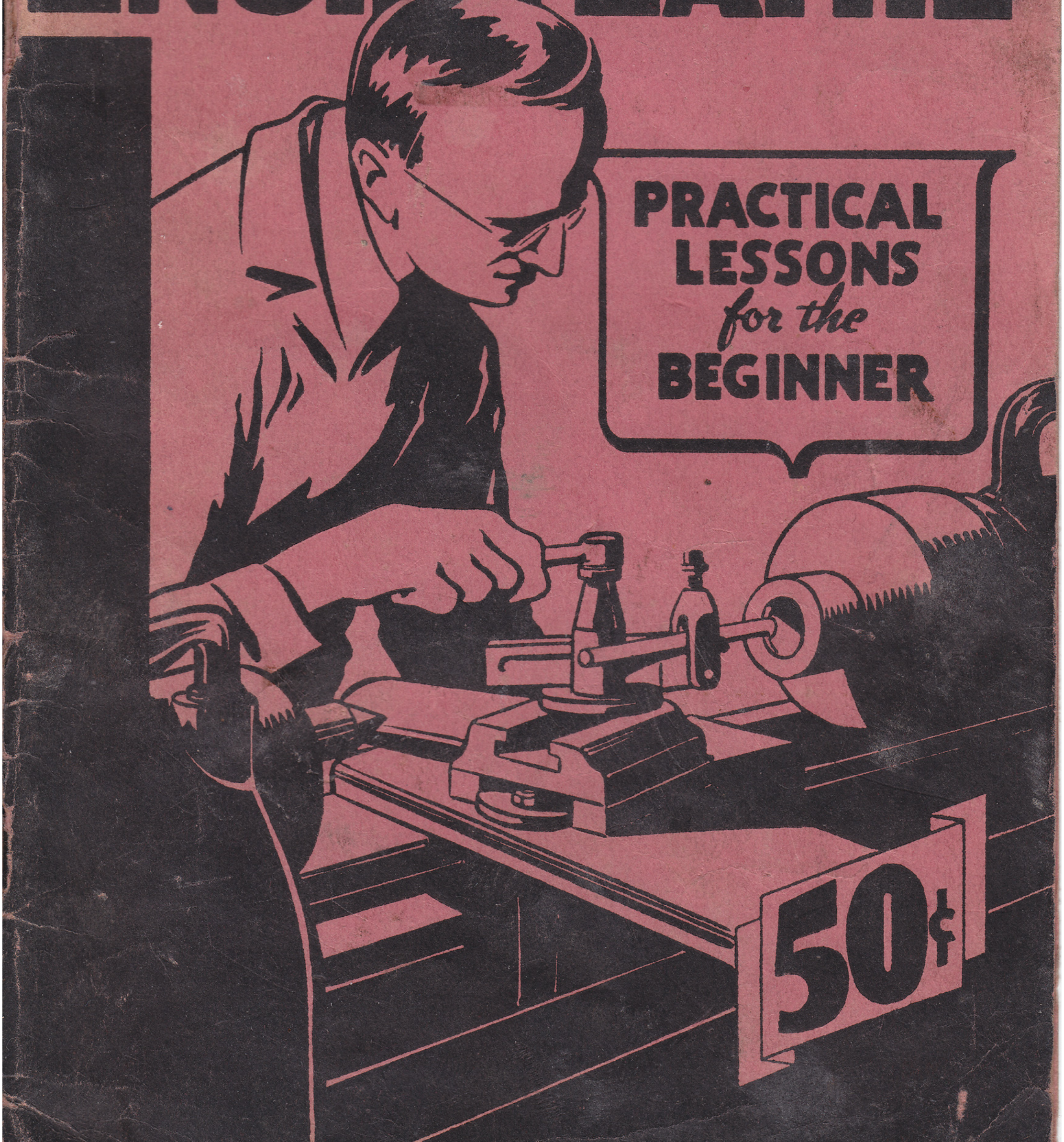
OPERATING an ENGINE LATHE



OPERATING AN ENGINE LATHE

By

RAY S. LINDENMEYER

Assistant Professor of Industrial Engineering
Northwestern Technological Institute

Published and Copyright, 1944, by
SCIENCE AND MECHANICS
154 East Erie Street, Chicago 11, Illinois

The Machinist in America Today

NEITHER America's ability to produce guns, ammunition, tanks, airplanes and battleships in volume, nor the present industrial era itself could possibly exist without the machinist and machine tools. Every new mechanical device or improvement of existing devices in the past one hundred years is heavily indebted to the machinist and his tools. Without them neither today's war nor tomorrow's peace can be won.

Metal parts requiring precision measurements, in the manufacture of any product, could not be as accurately made today as they are without the use of machine tools by expert machinists. They alone can produce parts with the close tolerances now specified.

With the nation fighting for its very existence, it is of the utmost importance that every person not on the fighting front and possessing mechanical ability should develop that ability by study and practice. This applies especially to the younger men and women who are working for the first time with machine tools in the war factories of our country.

To aid these beginners is the purpose behind the preparation and publication of this home study course on "Operating an Engine Lathe." The editors of "Science and Mechanics" chose the engine lathe for this study course because it is the universal machine tool in modern industry.

THE author of this course, Mr. Ray S. Lindenmeyer, has had eight years of practical shop experience with such concerns as Caterpillar Tractor Company, American Can Company, Metal Stamping Corporation, Rawls Manufacturing Company and others. He has served his apprenticeship as a tool maker and has worked as master mechanic, setup man, line foreman, inspector and layout man, experimental designer, draftsman and engineer.

Besides this practical work, Mr. Lindenmeyer is now in his tenth year as an instructor of shop practice. His teaching experience includes private school, high school, and junior college work in industrial arts and engineering. He also was director of industrial education at the Waukegan Township High School for a year and a half. He is now assistant professor of industrial engineering, in charge of engineering shop instruction, at Northwestern Technological Institute, Evanston, Ill.

In the war effort, Mr. Lindenmeyer has been the supervisor of a large national defense shop instructional program at Waukegan and was also in charge of civilian instructors of shop and related subjects in the trade training section of Great Lakes Naval Training Station.

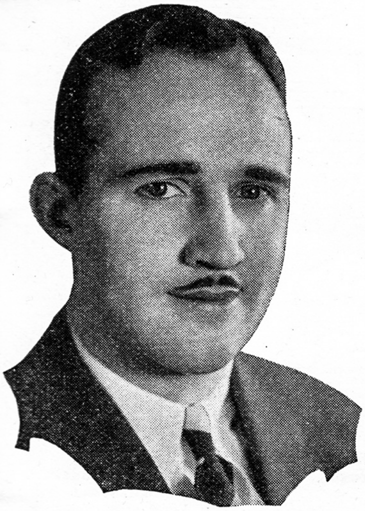
Mr. Lindenmeyer received his Bachelor of Science degree from Bradley Polytechnic Institute and his Master's degree from Northwestern University. He has studied at Carnegie Institute of Technology and has taken many evening and extension courses. To keep abreast of current developments in his field, he is a member of the American Vocational Association, American Society of Mechanical Engineers, and other such organizations.

Operating an ENGINE LATHE

Practical Lessons for the Beginner

By RAY S. LINDENMEYER

**Assistant Professor of Industrial Engineering
Northwestern Technological Institute**



RAY S. LINDENMEYER

More men are now becoming machine tool operators than at any other time in the history of the nation. To offer definite and practical aid to these men, this series of lessons on operating an engine lathe has been prepared by SCIENCE AND MECHANICS.

The aim is to give the beginner, in condensed form, the facts of science and general information that underlie successful lathe operation. The course is chiefly intended to supply a background which will make the operator resourceful and self-reliant and enable him to meet successfully new situations and demands made on him. The editors believe that it is possible for him by conscientious study and effort to merit promotion and to be an aid in the war tasks on the home front.

These lessons deal with the essential things that the beginner should know about the engine lathe itself, its attachments, and the tools used in lathe work. There will follow instructions on what the operator should be able to do, and beginning on page 54 will be found easy projects for him to work out.

Getting Acquainted with the Engine Lathe

LESSON 1

MASTER tool of them all, in today's metal-working shops, is the modern engine lathe. Far and away it is the oldest, the most adaptable, the most universally used, and therefore the most important of all machine tools made.

The lathe originated long before the planer, the milling machine, the grinding machine, the automatic screw machine, and the gear-cutting machine. Indeed, practically all of these machines were developed from the same basic design and operating principles that underlie engine lathe design and use. The engine lathe may well be called "the daddy of them all."

Likewise, America's vast industrial progress during the nineteenth and twentieth centuries would have been quite impossible without the engine lathe. It made possible the building of steamboats, steam and electric locomotives, electric motors, every kind of power-driven farm machinery, road-building and excavating machines, tractors, automobiles, and scores of other machines that do the heavy and even the light work of modern life.

There are many kinds of lathes: the wheel lathe, axle lathe, pulley lathe, jeweler's lathe, toolmaker's lathe, turret lathe, etc. Almost every metal-working and wood-working trade has a lathe specially designed for the kind of work done by that trade. For general work certain more or less standard designs of the engine lathe are used. These are: back-geared lathe, geared-head lathe, turret lathe, etc. Each of these will be defined later, as they come up in these lessons.

The term "engine lathe" deserves a word of explanation. Originally, of course, the only lathes made were operated by hand or perhaps foot power. Later when some form of mechanical power was applied to the machine, it was called the "engine lathe." And that term has lived on through the years.

Originally designed for making cylindrical surfaces, engine lathes have been steadily improved until they can also be used for drilling, reaming, boring, even making flat surfaces, and for cutting screw threads. They can also be used, with special attachments, for milling and grinding where other machines made for these two operations are not available.

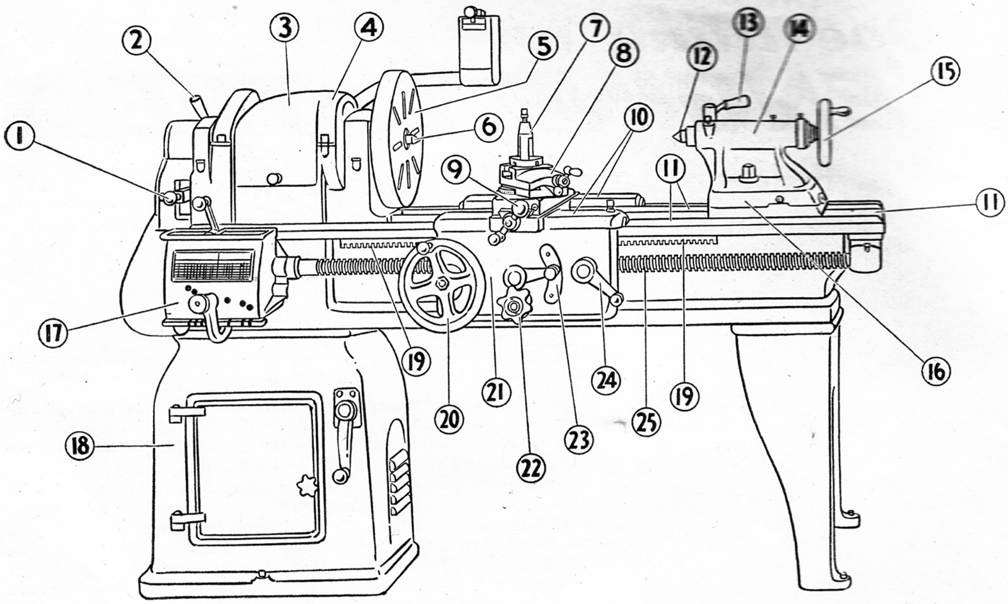


FIG. 1. A FLOOR TYPE BACK-GEARED ENGINE LATHE

1. Feed reverse lever
2. Back gear lever
3. Cone pulley cover
4. Headstock
5. Faceplate
6. Live center
7. Tool post
8. Compound rest
9. Cross-slide screw
10. Saddle of carriage
11. Bed
12. Dead center
13. Tail-spindle lock
14. Tailstock
15. Tailstock handwheel
16. Tailstock base
17. Gear box
18. Motor drive in leg
19. Carriage rack
20. Hand feed
21. Apron of carriage
22. Clutch knob
23. Feed change lever
24. Half-nut lever
25. Lead screw

In getting acquainted with an engine lathe, the beginning operator may well start out with the back-geared lathe, because it is the simplest form, the oldest one, and also because many of these lathes are in use throughout the country.

The new operator should first learn the names of the various parts of a lathe before starting to operate it. The main parts are shown in Figs. 1 and 2. Study both illustrations closely. These names should become part of the vocabulary of the operator.

Besides knowing the names of these parts, it is also necessary to know their functions. While these operations are general, they apply to most all makes and types of lathes. The ambitious lathe operator should obtain information from the manufacturer of the lathe that he is or will be operating. In this way he can get specific details of that particular machine.

Now let us look more closely at the construction features of a lathe and their functions.

The *bed* is the main heavy framework supported by metal legs or mounted on a bench. This is usually designed so as to resist stresses set up by machining. The upper surface has V's planed on it for aligning various other parts.

The *headstock* is the unit that rests and is fastened securely on the left end of the bed. It is this part that causes the work to revolve while it is being processed and acts as a support for chucks, faceplates, and the live center. Also, in the headstock are gears which transmit power to the feeding mechanism.

The *tailstock* also rests on the bed but at the right end and is movable to take care of variable length pieces of work. It supports work being turned between centers and may also be used to hold drill chucks, reamers, drills, and special tools for performing these operations while the work is being turned.

The *carriage* is the mechanism which rests between the head and tailstock on the V-ways of the bed. It is movable and serves to carry the cutting tool which is held in the tool post.

The *leadscrew* is used to transmit power to the carriage either through power feed clutch or through the split nut when cutting threads.

The *live center* fits in the spindle of the headstock and is used to support work at the end. It is usually soft and can be machined so that it runs absolutely true.

The *dead center* fits in the tailstock spindle, supports the outer end of work turned between centers and is hard. It has a 60 degree angle.

The *cone pulleys* are for speed variations when turning various size pieces of work as well as material of varying machinability. On the geared head lathe these variable r.p.m.'s are ob-

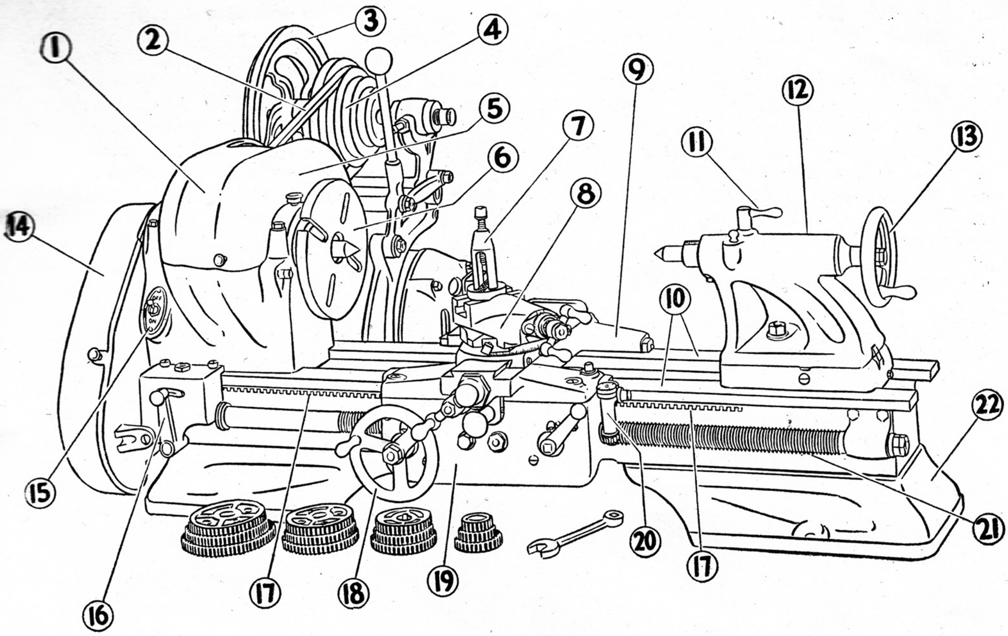


FIG. 2. A BENCH TYPE BACK-GEARED ENGINE LATHE

1. Headstock
2. Power shaft
3. Pulley (to motor)
4. Pulleys to back gears
5. Back gears (enclosed)
6. Faceplate on spindle
7. Tool post
8. Compound rest
9. Saddle of carriage
10. Bed
11. Tail-spindle lock
12. Tailstock
13. Tailstock handwheel
14. Gear train (enclosed)
15. Off-and-on switch
16. Reverse lever
17. Carriage rack
18. Hand feed
19. Apron of carriage
20. Threading dial
21. Lead screw
22. Lathe base

tained by the manipulation of the levers.

The *gear change box* serves to allow minute variations in the feed for the travel of the carriage as used in turning or in screw cutting.

The *back gears* are used to reduce speed and to increase the power when necessary for turning large pieces of work or tough pieces of material, and sometimes on special operations.

The feed *reversing lever* and gears are arranged to permit control of the direction of feed either to the right or left longitudinally, and in or out of the cross feed.

The *saddle* is part of the carriage resting on the V-ways which causes the carriage to move in alignment with the lathe centers.

The *apron,* also part of the carriage, contains the gear mechanism, clutch, feed control lever, and split nut.

The *cross-slide* rests on the saddle, has an in and out motion that can be controlled by power or by hand. It can be set to minute variations and the *compound rest* is fitted to the top of the cross-slide and can be set at any angle. It is used for turning steep tapers or angles.

The *tool post* fits into a T-slot in the compound rest and is used to hold the tool holder.

The *half-nut lever* is used only when performing threading operations. It is so arranged that the feed mechanism of the lathe must be in neutral position before it can be engaged.

The *clutch knob* controls the feed mechanism for longitudinal and cross-feed transmission.

The *feed change lever* is the selector for longitudinal or cross-feed motion direction.

The *faceplate* fits on the spindle and is used to transmit motion through the lathe dog to the work that is being processed between centers.

The motive power for the lathe may be furnished from a motor fastened directly on the machine or from a countershaft mounted on a pedestal to the rear of the machine or from a lineshaft and countershaft mounted above the machine.

After the operator has become acquainted with the general construction features and function of these parts, he may then attempt to operate them. Some things which he should be able to do as a tryout experience are as follows:

Cause the machine to operate at each of its various speeds. If the machine is not equipped with a plate showing these r.p.m.'s, he should obtain them through the use of a speed indicator or tachometer, or calculate them from the diameters of pulleys and speed of the motor at the source. As part of the speed control, the back gears must be placed in and out of position by the operator. This lever should *not* be moved when the machine is in motion. The operator

should practice this until he can do it with ease.

The centers, both live and dead, should be removed, replaced in proper position and properly aligned by the operator. This is one of his first duties each time before using the lathe.

The faceplate, or chuck, should be removed and replaced without damaging the machine and with safety to the operator. These parts should screw on freely and all the way up to the shoulder. The secret in doing this is to keep the threads clean and apply a drop of oil before replacing these accessories.

The operator should next become familiar with the gear change box and be able to set the feeds by manipulation of the various levers. Some lathes do not have the gear change box and use a series of gears or step pulleys to accomplish feed variations. As part of the feed control operations, the operator should cause the carriage to travel longitudinally both to the right and left and the cross-slide in an in and out direction. He should also become familiar with the operation and action of the half-nuts that are used when performing threading operations.

No work should ever be attempted by the operator until he is familiar with his lathe.

Special Accessories Are Necessary

LESSON 2

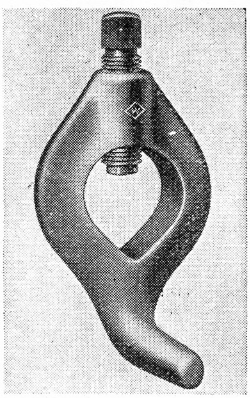


Fig. 9. Standard lathe dog.

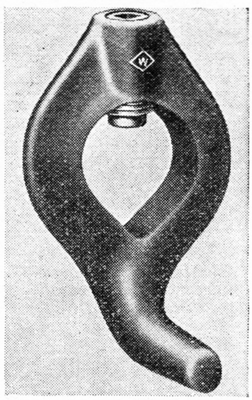


Fig. 10. Safety lathe dog.

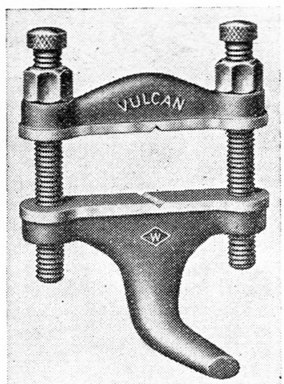


Fig. 11. Clamp lathe dog.

SPECIAL accessories used in connection with the lathe (as explained in Lesson 1) facilitate its operation and are necessary: They are as follows:

The *lathe dogs.* Figs. 9, 10, 11, are typical examples. These are fastened securely to one end of the work and serve as a means of transfer of rotation from the headstock through the driving or faceplate to the work. These come in various sizes and shapes as shown. The operator should select a dog with an opening just slightly larger than the piece of work to which it is to be attached. He should also make certain that the tail of the dog fits freely into the groove of the driving plate.

Fig. 12. Tool holder with cutter bit.

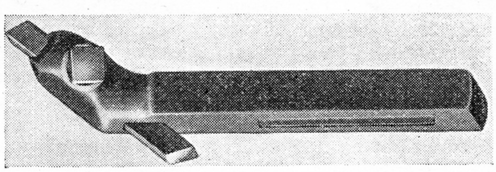


Fig. 13. Lefthand tool holder.

Fig. 14. Righthand tool holder.

Tool holders, Figs. 12 to 18, are considered more or less standard for holding tool bits. This type permits holding the tool bit securely and gives it adequate support. Each of the various shapes and types is used as required for the various classes of work that cannot be readily accomplished by one single shape. They come in various sizes for use on the various machines and tool bits. More will be said later in the lessons on lathe tools and their application and on how to grind and set up the tool for turning.

Lathe chucks, Figs. 19 and 20, are used to support work that cannot be mounted between centers while it is being machined. There are three main types and several minor variations. Those principally used are:

The *independent chuck,* Fig. 19, which usually has

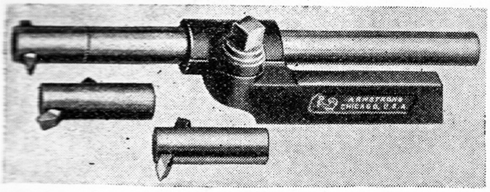


Fig. 15. Boring tool.

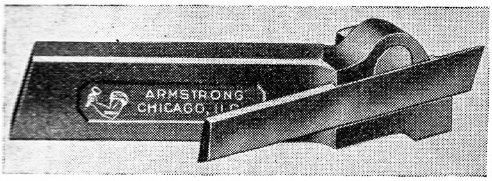


Fig. 16. Cutting-off tool.

Fig. 17. Formed threading tool.

four jaws, each one operating independently of the other. This type is used for chucking round, square, and odd-shape pieces. Work can be made to rotate in either concentric or eccentric motion as desired.

The *universal chuck,* Fig. 20, usually has three jaws that move simultaneously and automatically centers the work. It is usually of the three-jaw variety but is sometimes made with either two or four jaws. This type of chuck has two sets of jaws, one for external chucking and the other for internal chucking. It is very quick in operation and is used for chucking both round and other shapes depending on the number of jaws. The accuracy of this type of chuck is impaired by rough handling and the chucking of rough castings.

The *combination chuck* which may have two, three, or four jaws in which any jaw may be moved singly or all of them in unison. This type of chuck lends itself to production on uniformly shaped pieces.

It is very important that the operator select the proper type and size chuck for the work he is to do. If a piece of work is chucked only with the edge of the jaws repeatedly, it will spring the chuck and render it useless. Chucks are expensive pieces of equipment and should be kept cleaned and properly stored when not in use.

Mandrels, Fig. 21, are hardened and ground

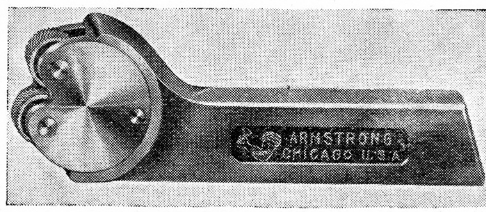


Fig. 18. Knurling tool.

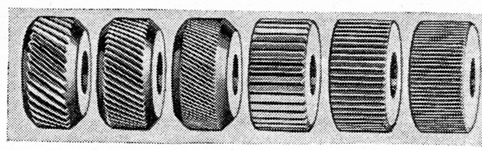


Fig. 18-A. Knurling cutters.

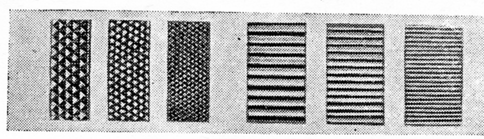


Fig. 18-B. Knurling pattern produced.

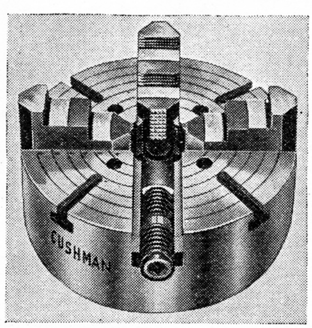


Fig. 19. Independent chuck.

cylindrical shapes with center holes at the ends. They are used to support work which previously had a bored or reamed hole in it, and requires additional turning operations. The plain mandrel has a taper of .006″ per foot and has its size stamped on the large end. The small end is slightly under size to permit free entering of the hole. A drop of oil should be placed on the mandrel before putting it into place. Other varieties of lathe mandrels are of the expanding type which is used when a hole may be slightly over or under a standard size. It may also be used for standard size holes. Another special mandrel is the gang or nut

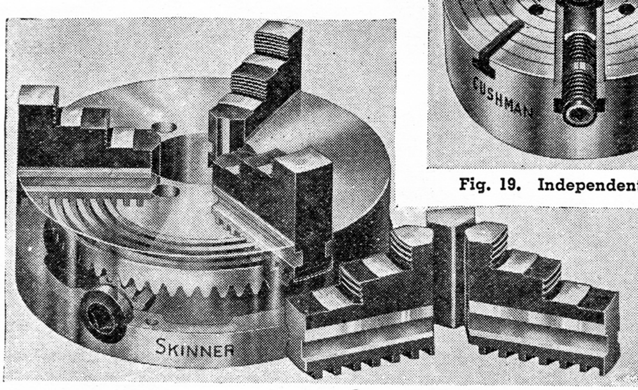


Fig. 20. Universal chuck.

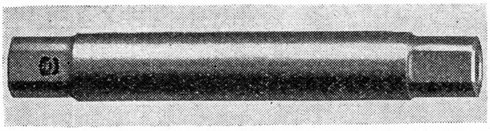


Fig. 21. Steel lathe mandrel.

mandrel and is used on production of many similar parts where the holes may be of variable sizes within a small tolerance. It does not rely upon friction in the hole for its driving power. In removing a plain mandrel from a piece of work, be sure that it is removed in the proper direction. Plain lathe mandrels may be purchased in a variety of standard dimensions and special ones may be made up by the operator as the need arises.

If the number of parts to be turned is not large, the lathe operator often makes his own mandrel. If many parts are to be made, it is better to use a well made hardened mandrel.

Other Attachments Often Required

LESSON 3

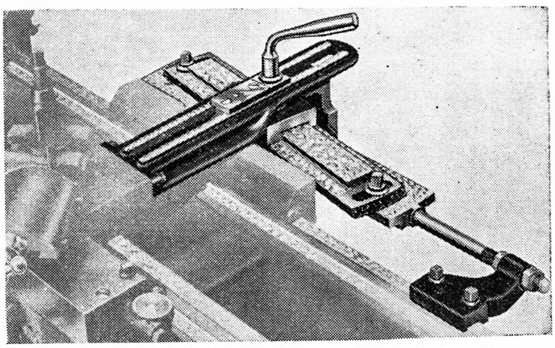


Fig. 22. Taper cutting attachment.

THE following attachments are more or less considered special and any or all of them can be purchased according to the needs. The toolmaker's lathe sometimes comes equipped with many of these devices.

The *taper attachment*, Fig. 22, is attached by a bracket at the back of the lathe bed and is arranged to operate the cross slide in such a manner as to produce a taper. The slide bar, which is adjustable, is graduated on one end in degrees, and on the other end in taper per foot in inches. Its adjustment is very simple and can be set up very quickly. Duplicate tapers can be cut quickly even on pieces of different lengths. It can likewise be used in taper boring. The particular advantage of this method of cutting tapers over the tailstock set-over method is that the centers need never be taken out of alignment, the bearing surface of the centers are unaffected, and the variation in length need not be compensated for.

The *steady rest*, Fig. 23, is used to support the work during turning, boring, and threading operations when these operations cannot be performed in any other way without undue chatter and vibration. By using the steady rest, work of small diameter that is quite long can be turned accurately at a much faster rate of speed. When boring, it is necessary to either chuck the other end of the work or tie the dog securely to the driving plate. The steady rest is clamped to the bed and remains stationary during all turning operations.

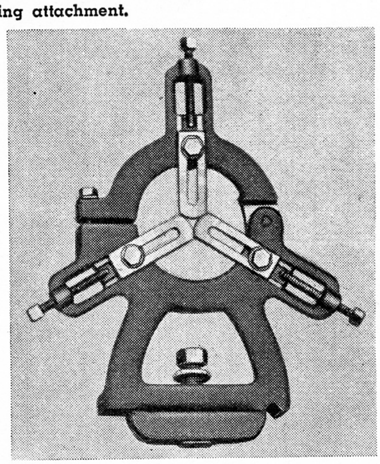


Fig. 23. Steady rest.

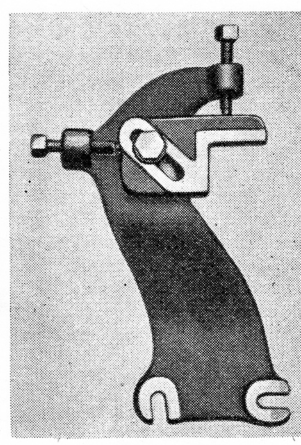


Fig. 24. Follower rest.

The follower rest, Fig. 24, like the steady rest, is used to support a round piece of work against the action of the cut being taken. It, however, is bolted to the carriage and moves with the tool. The follower rest is very useful when machining small delicate spindles and especially when cutting threads on long slender shafts. Oftentimes both the steady rest and the follower rest are used at the same time.

The *micrometer stop*, Fig. 25, may be clamped on to the front of the lathe bed in such a manner as to act as a guide when machining up to a shoulder. In this way pieces may be

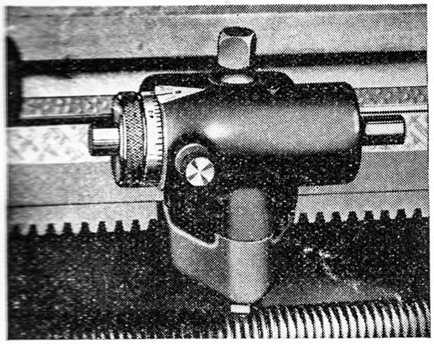


Fig. 25. Micrometer carriage stop.

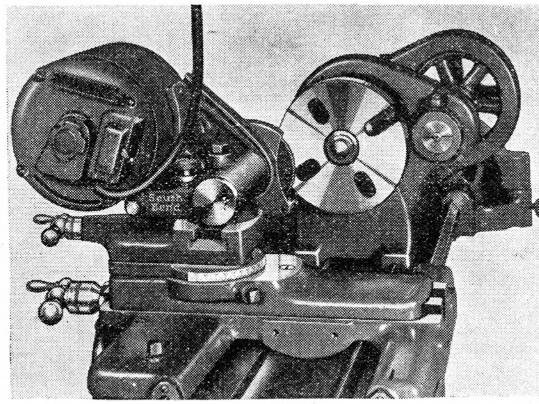


Fig. 26. External grinding attachment.

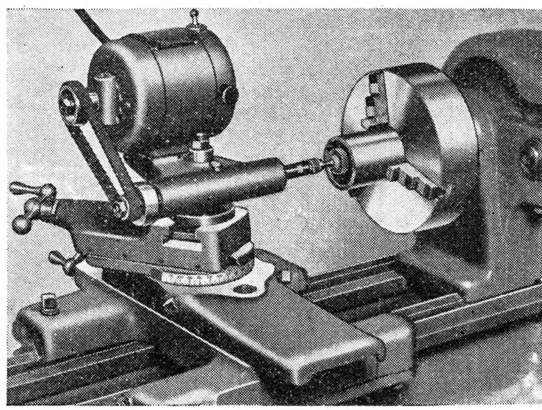


Fig. 27. Internal grinding attachment.

faced to exact measurements. It is very convenient for production operations along similar lines and is included as standard equipment on most toolroom lathes. The fact that it has a micrometer adjustment makes it useful on precision measurements. The carriage must never be allowed to run up to the stop automatically but should be stopped a short distance from it and the remainder fed by hand.

The *tool-post grinders,* Figs. 26 and 27, may be attached to the lathe in such a manner to perform external and internal grinding operations such as grinding lathe centers, reamers, cutters, drill bushings, and cylindrical-straight and tapered shafts and holes. There is one disadvantage, however, in grinding on the lathe, and that is that the abrasive and small particles of metal find their way into the bearings and bearing surfaces of the machine and eventually affect its accuracy. To counteract this to a great extent, if this method must be used, it is best to protect such bearings by covering them with an oil-dampened cloth. This will collect most of the dust and grit.

The *turret attachment,* Fig. 28, is a special accessory that replaces the tailstock assembly. It is used only when quantity production is desired or when conditions warrant its use. In the tail turret, a succession of tools can be caused to come into operation as needed, such as drilling, boring, reaming, tapping, and counter boring. External turning operations may be going on at the same time as the turret tools are in operation.

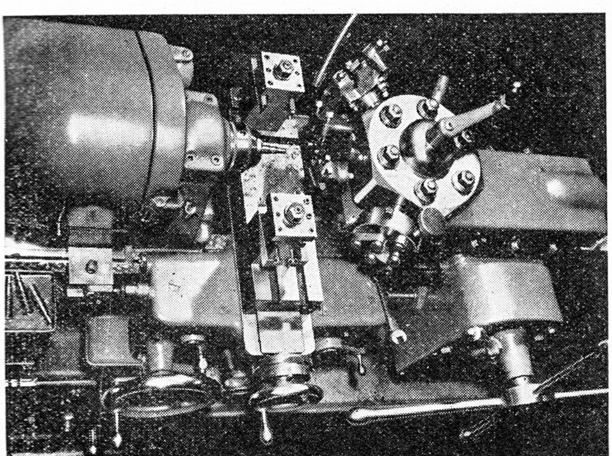


Fig. 28. Turret attachment on bed.

The *thread dial indicator,* Fig. 29, is a small device attached to the carriage and geared with the lead screw. With this device the operator can engage and disengage the half-nuts at the beginning and the end of the thread to be cut, thereby saving considerable time, especially when cutting long screw threads. A few simple rules for its operation are as follows:

1. For all even numbered threads, the half-nuts may be closed on any line on the dial at any time whether that line is numbered or not.

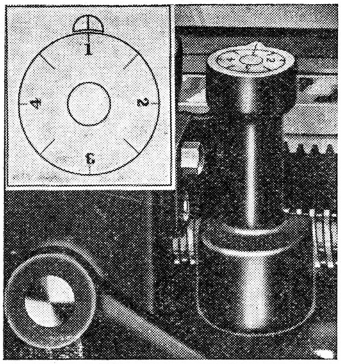


Fig. 29. Thread dial indicator.

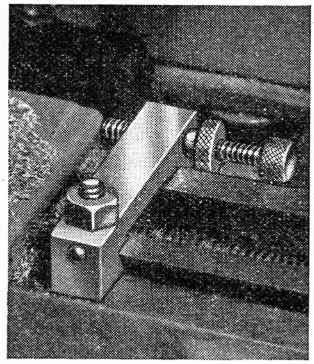


Fig. 30. Cross-slide stop.

2. For all odd numbered threads, the half-nuts may be closed only on any numbered line on the dial at any time.

3. For those threads involving one-half of a thread in each inch, such as 11½, the half-nuts must be closed on opposite numbered lines only such as 1 and 3 or 2 and 4.

4. For quarter threads, it is necessary that the dial be on the same number as was originally used at the start of the thread.

It is not necessary to use the thread dial indicator at all if one is producing a thread which is equal to or an exact multiple of the number of the threads on the lead screw. On threads cut only at short distances, the split nut need not be disengaged, instead the lathe may be caused to run in reverse to bring the tool back to the starting point.

The *cross-slide stop,* Fig. 30, is especially valuable in thread cutting because the tool must be withdrawn at the end of the cut.

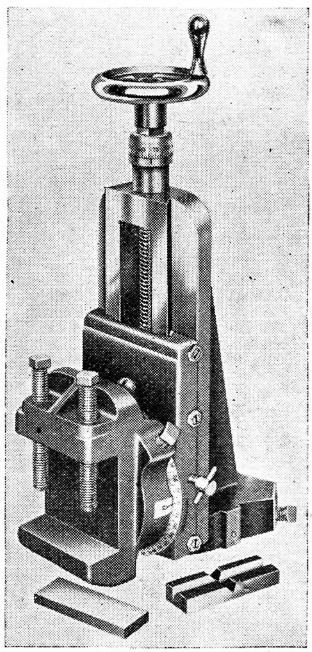


Fig. 31. Milling attachment.

When the tool is brought back to the starting place, it is only necessary to turn the cross-feed screw in as far as the stop will permit. The cross-slide stop then serves in place of remembering the number on the micrometer collar on the cross-feed screw and thus saves operating time. This method is used when the compound rest is placed at 30 degrees and all feeding of the tool is done at this angle. The stop may also be used when turning duplicate pieces and again saves operator's time in remembering where the tool had previously been set.

The *milling attachment,* Fig. 31, is special and can be used to convert the lathe quickly into a milling machine capable of performing many small and varied milling operations. It is primarily used when using small cutters for squaring work, cutting keyways, slots, grooves, T-slots, and dovetails and may also be used for some facing operations. It is attached on the cross slide in the place of the compound rest and in this way can be moved in an in-and-out or longitudinal direction. It can also be adjusted up and down by hand feed and locked in these various positions and the attachment vise may be swiveled at any desired angle. The milling cutter may be held in a collet, taper, or chuck.

The *gear cutter attachment,* Figs. 32 and 33, when used on the lathe, will aid in producing

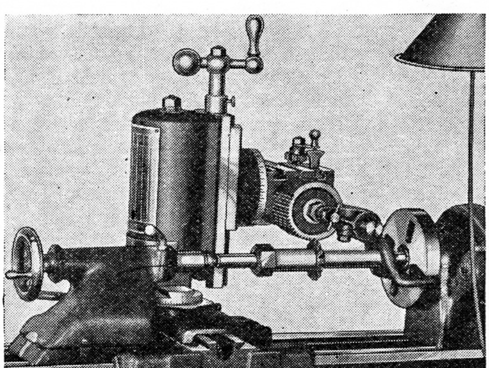


Fig. 32. Gear cutting attachment.

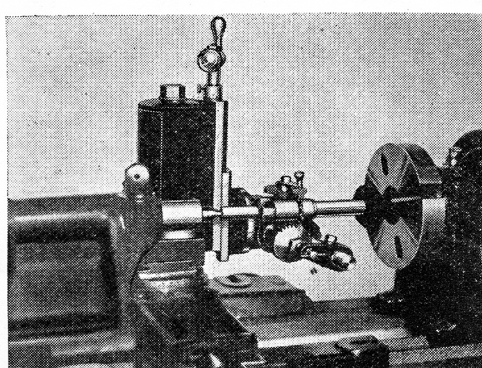


Fig. 33. Cutting a gear on a lathe.

spur and bevel gear-teeth. Like the milling attachment, it fits on the cross slide and may be used also in graduating and milling keyways, angles, spines, and other index head milling operations. The dividing head principal is similar to those used on regular milling machines.

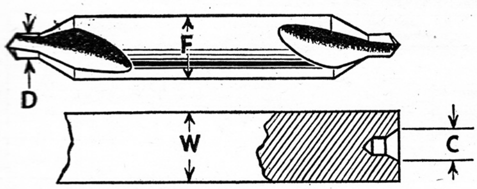
With the aid of these attachments the operator has a universal machine tool with which he can completely machine a very complicated product. Because some of these accessories are very rarely used, they will not be found on every lathe, especially the milling attachments. More information on how to use these attachments will be given in future lessons.

Lathe Center Holes Must Fit Your Work

LESSON 4

CENTER holes for work to be done between centers on a lathe should be sized to fit the work. Center holes may be drilled in a number of ways: in the drill press, centering machine, holding the work in a chuck in the lathe itself, causing the work to rotate and placing a center drill in the tailstock spindle, or placing a center drill in the headstock spindle and supporting the work at one end by the tailstock spindle. Fig. 34 gives the approximate size of center holes when turning work from 3/16″ to 4″ in diameter.

FIG. 34. SIZE OF CENTER HOLE FOR 3/16 IN. TO 4 IN. DIAMETER SHAFTS.



Diameter of Work W	Large Diameter of Counter-sunk Hole C	Diameter of Drill D	Diameter of Body F
3/16 in. to 5/16 in.	1/8 in.	1/16 in.	13/64 in.
3/8 in. to 1 in.	3/16 in.	3/32 in.	3/10 in.
1¼ in. to 2 in.	¼ in.	1/8 in.	3/10 in.
2¼ in. to 4 in.	5/16 in.	5/32 in.	7/16 in.

Finding the Correct Speed and Feed

LESSON 5

THE correct speed and feed at which to operate a lathe is often left to the operator's experience or judgment. However, a few rules will act as a guide or basis to work from. The operator may find through experience that it may be possible to increase the speeds and feeds or he may find it necessary to decrease them. A very simple formula for figuring the approximate r.p.m. at which a piece of material should operate is as follows: r.p.m. equals cutting speed multiplied by 4, divided by the diameter in inches. This formula, $\frac{\text{C. S.} \times 4}{\text{D}''}$, is derived from figuring the circumference of the work and the amount of material removed in one minute.

In order to use this formula, it is necessary to set up a few of the machinability figures for various kinds of material. First, let us assume that we are using high speed tools and that we are taking roughing cuts. When using carbon tool bits, these figures must be cut in half. When using carbide tool bits, the figure given may be doubled. For finishing speeds, using high speed tool bits, the figures may also be increased 50% to 100%. Assuming also that material is removed in proportion to the size of the machine as far as depth of cut is concerned, then the following speeds, (c.s.) in feet per minute (f.p.m.), may be used.

1—Mild steel, 100 feet per minute.
2—Cast iron 75 feet per minute.
3—Tool steel, 50 feet per minute.
4—Brass, aluminum, 200 feet per minute.

Let us attempt a typical example: A piece of cold roll steel 2″ in diameter is to be turned down. Substituting in the formula, assume that the cold roll is mild steel. Therefore, it can be machined approximately 100 feet per minute, so we take 100 × 4 which gives us 400 and divide this by 2. The answer is, therefore, 200 r.p.m. the speed to use when performing the roughing operations. As the piece gets smaller this speed may be increased.

The usual tendency of the lathe operator is to underestimate the capacity of the machine with respect to speeds and feeds. A great deal of difficulty is due to the operator's misunderstanding with reference to the cutting action of tools. This will be discussed in a later lesson.

Feeds, necessary to all turning, are underestimated even more often than speeds. The cut and try method is used by most operators, taking into consideration the type of finish and accuracy necessary on the part that is to be machined. A good cut-and-try method for both speed and feed after what few simple calculations are made, is to increase the feed and sometimes the

speed to the point where the machine tends to slow down, or the tool breaks down in service, considering of course the type of finish desired. When this point is reached, either the speed or feed or both may be decreased slightly to the point where the desired results are obtained.

As a rough estimate, one may say that roughing cuts may be made at moderate speeds with deep cuts and fast feed while finishing cuts are made at a relatively high speed with shallow cuts and slow feed.

The depth of the cut for roughing varies from 1/16 to 1/8 and even more on larger stock with larger machines, and finishing cuts are usually 1/64 of an inch or less in depth but very seldom less than .005″ as this does not allow enough material to form the desired finish.

Roughing feeds vary from 1/32 to 1/16 or more travel per revolution of the work and finishing feeds may be 1/64 or less per revolution.

Speeds are governed by cone pulleys or variable speed mechanisms while the feed may be varied either through step pulleys on the older machines or by geared transmissions on the new machines.

The operator can profit by his experience with regard to speeds and feeds and it will pay him well to keep a note-book.

Setting All Your Cutting Tools Accurately

LESSON 6

THE cutting tool shapes, Fig. 35, are those that are most generally used in performing the various turning operations on the lathes. Minor variations from these may be necessary under certain circumstances.

The important things about tool bits are their angles of clearance and rake. These vary with the machinability of the material. In general, for those materials which are dealt with by the beginner, a side clearance angle of 8 degrees and a front clearance of 8 to 10 degrees will usually be adequate. Too much clearance will result in a weakened condition of the cutting edge while insufficient clearance will prevent the tool from functioning. The side and back rake facilitates free cutting. It is also variable for the various kinds of material. The harder the material, and for cast iron and hard bronze, little or no rake is required, while the softer the material the more this rake may be increased.

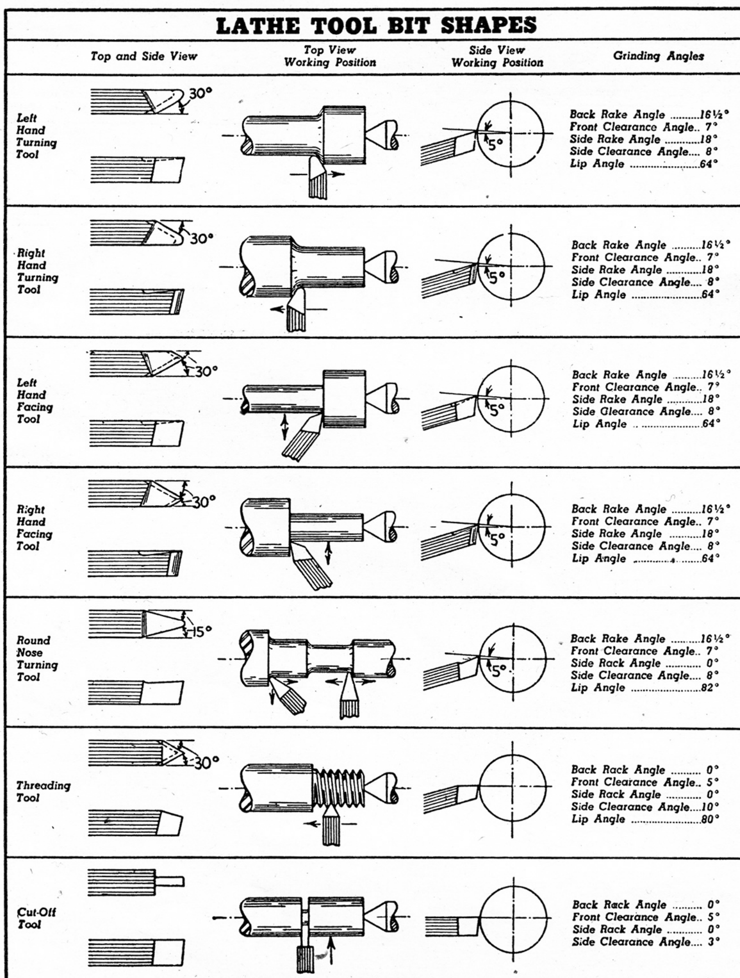


Fig. 35. Bit shapes, working positions, and grinding angles.

The cutting edge of the tool must be honed after grinding. This will improve the quality of the finish on the work and many times lengthens the tool life as much as 50% to 100%. More will be said about tool bits in the lesson on grinding cutters with reference to angles of clearance and rake and speeds and feeds. Make it a point to study Fig. 35 closely, and to understand back rake, front clearance, side clearance, and side rake.

Keep a Good Steel Rule in Your Pocket

LESSON 7

THE steel rule, Fig. 36, comes in a variety of shapes and sizes with special features, but the method of using and reading it are all alike. It is necessary that the operator acquaint himself with this procedure by following the instructions given below. By studying the drawing, Fig. 37, he should become proficient in this operation.

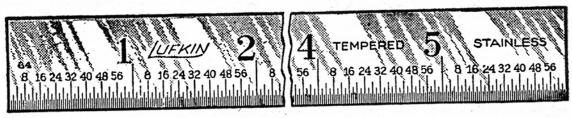


Fig. 36. Steel rule.

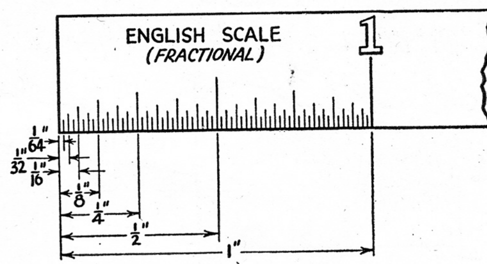


Fig. 37. How to read a scale.

Most fractional dimensions given on prints must be held to a plus or minus tolerance of $\frac{1}{64}''$. A good machinist never says $\frac{4}{8}''$, $\frac{8}{16}''$, or $\frac{48}{32}''$, although such expressions as $\frac{1}{64}''$ over $\frac{1}{2}''$ or $\frac{1}{32}''$ less $\frac{3}{4}''$ are very commonly used. Other terms used by the machinist are a scant $\frac{1}{8}''$ or a full $\frac{1}{16}''$, and a half 64th. In reading a scale, the user should find the nearest significant graduation, then add or subtract to obtain the fraction to its nearest 64th.

A good rule of reliable make should be the property of every mechanic. One of 6″ length of the flexible variety, graduated in 64ths is frequently recommended. If possible measurements should not be made from the end of the rule but rather from the first significant whole number. Additional use and application of the scale in measuring will be discussed in future lessons of this series.

Among other scales used are the English decimal scale and the metric scale.

Outside and Inside Calipers Help You Do Better Work

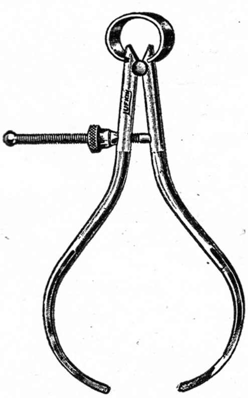


Fig. 38. Outside caliper.

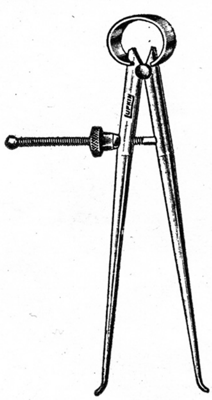


Fig. 39. Inside caliper.

THE caliper is a useful tool in the hands of a good operator in determining approximate measurements of external and internal work. Care in its use is necessary to obtain correct readings. Calipers may be had in a variety of sizes and with special construction features.

LESSON 8

It is possible to check dimensions within a few thousandths of an inch through the proper uses of these tools.

To set an outside caliper, Fig. 38, to a diameter with the use of a scale, the procedure is as follows: Hold the scale in the left hand and the caliper in the right hand, using the thumb of the left hand to keep the one leg of the caliper from slipping off the scale. Adjust the caliper to the dimension required with the aid of the forefinger and the thumb of the right hand shown in Fig. 40.

When using the outside calipers to check a piece of work for size, the caliper is held in a vertical position as in Fig. 42, with the legs at right angles to the axis of the work. The calipers are then adjusted to the place where their own weight will allow them to pass over the diameter of the piece.

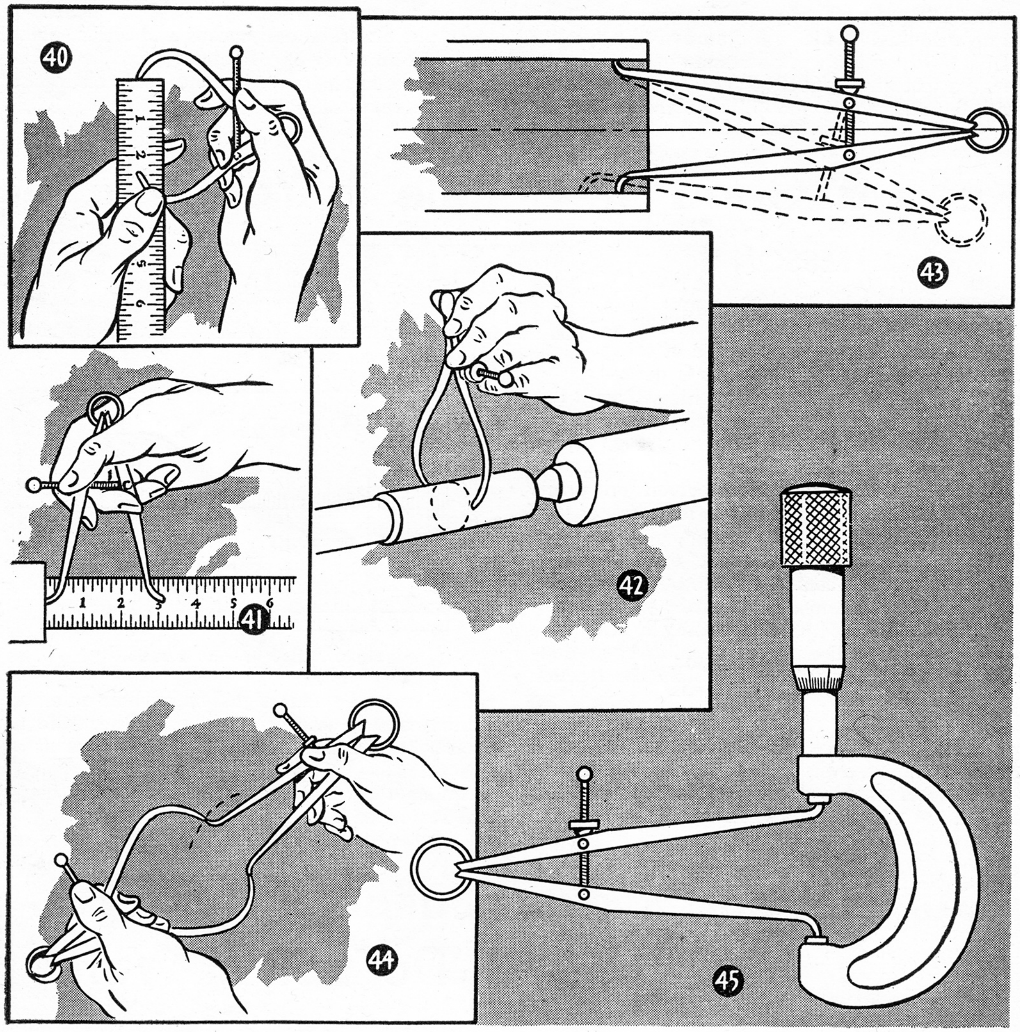


Fig. 40. Setting outside caliper to scale.

Fig. 41. Setting inside caliper to scale.

Fig. 42. Measuring with outside caliper.

Fig. 43. Measuring with inside caliper.

Fig. 44. Transferring measurement from inside to outside caliper.

Fig. 45. Setting inside caliper to micrometer reading.

The slight resistance is vitally necessary for accurate measurement and is referred to as "caliper feel." If the calipers are forced over the work it will result in inaccuracy as the legs will spring. Work should never be calipered while it is revolving, as inaccuracy is sure to be the result.

Inside calipers, Fig. 39, are used for taking internal dimensions of work such as bored or reamed holes. The inside caliper is best set to a scale, Fig. 41, by holding the scale against a flat machined surface. When boring to accurate dimensions, it is necessary to set the inside calipers to a micrometer reading, Fig. 45. The same "feel" that is used in taking this transfer measurement should be used in calipering the hole. When measuring the hole, Fig. 43, set one leg of the caliper to one side, pivoting the calipers in a vertical plane and adjusting until the other leg enters and the proper "feel" obtained. This takes careful adjustment, making certain that the caliper legs are across the largest diameter of the hole.

In some operations, it is necessary to transfer measurements from the outside to inside calipers and vice versa. To do this, it is necessary to hold one pair of the set of calipers in the left hand and the pair that is being adjusted in the right hand. When the proper feel is obtained, the transfer is complete, Fig. 44.

Decimal Equivalents to Be Memorized

LESSON 9

DECIMAL EQUIVALENTS

FRACTION	INCHES	M/M	FRACTION	INCHES	M/M
1/64	.01563	.397	33/64	.51563	13.097
1/32	.03125	.794	17/32	.53125	13.494
3/64	.04688	1.191	35/64	.54688	13.890
1/16	.06250	1.587	9/16	.56250	14.287
5/64	.07813	1.984	37/64	.57813	14.684
3/32	.09375	2.381	19/32	.59375	15.081
7/64	.10938	2.778	39/64	.60938	15.478
1/8	.12500	3.175	5/8	.62500	15.875
9/64	.14063	3.572	41/64	.64063	16.272
5/32	.15625	3.969	21/32	.65625	16.669
11/64	.17188	4.366	43/64	.67188	17.065
3/16	.18750	4.762	11/16	.68750	17.462
13/64	.20313	5.159	45/64	.70313	17.859
7/32	.21875	5.556	23/32	.71875	18.256
15/64	.23438	5.953	47/64	.73438	18.653
1/4	.25000	6.350	3/4	.75000	19.050
17/64	.26563	6.747	49/64	.76563	19.477
9/32	.28125	7.144	25/32	.78125	19.884
19/64	.29688	7.541	51/64	.79688	20.240
5/16	.31250	7.937	13/16	.81250	20.637
21/64	.32813	8.334	53/64	.82813	21.034
11/32	.34375	8.731	27/32	.84375	21.431
23/64	.35938	9.128	55/64	.85938	21.828
3/8	.37500	9.525	7/8	.87500	22.225
25/64	.39063	9.922	57/64	.89063	22.622
13/32	.40625	10.319	29/32	.90625	23.019
27/64	.42188	10.716	59/64	.92188	23.415
7/16	.43750	11.113	15/16	.93750	23.812
29/64	.45313	11.509	61/64	.95313	24.209
15/32	.46875	11.906	31/32	.96875	24.606
31/64	.48438	12.303	63/64	.98438	25.003
1/2	.50000	12.700	1	1.00000	25.400

Fig. 46. Decimal equivalents of fractions.

EVERY lathe operator is called upon to turn work to very close tolerances, usually stated in decimal dimensions. It is, therefore, very important that he commit to memory certain decimal equivalents. Fig. 46 lists by 64ths the decimal equivalent of all fractions from 1/64″ to 1″. It is not necessary that he know all of these, but a very simple method of determining them all may be obtained by committing to memory such decimal equivalents as 1/64″, 1/32,″ 1/8″, 1/4″, and 1/2″. If these are memorized, as you will see, it is very easy to obtain any decimal equivalent by adding or subtracting from a known figure. Constant usage of this method will add speed to an operator. However, when in doubt it is best to refer to a decimal equivalent chart.

Let us try the above method to see how it operates. First, we will assume the following:

1/64″ equals approximately ..... .016
1/32″ equals approximately ..... .031
1/16″ equals approximately ..... .062
1/8″ equals .... .125
1/4″ equals .... .250
1/2″ equals .... .500

Now if an operator needs to know what 31/64″ is equivalent to in decimals, he need only subtract .016″ from .500″, which is 1/2″ minus 1/64″, and his answer will be .484″, which is accurate to three places. Other examples might be finding a decimal equivalent for 25/32″ which is 1/32″ more than 3/4″ or .031″ more than .750″, which is equal to .781″. Another example might be 5/8″ which is .125″ more than 1/2″, or .625″. Similar examples may be practiced by the operator until he gets proficient with this method.

Be an Expert in Using the Micrometer Caliper

LESSON 10

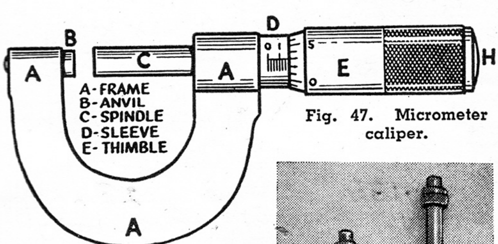


Fig. 47. Micrometer caliper.

ASSUMING that the operator is acquainted with decimal equivalents and has the ability to write and calculate them, it is a simple matter to learn to use and measure with micrometer calipers.

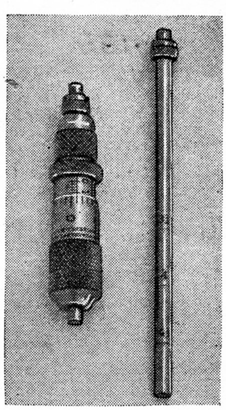


Fig. 48. Inside micrometer caliper.

Micrometer calipers, Fig. 47, come in a variety of sizes and are made for taking both internal and external measurements. Special types of micrometers are made for special application such as those used for measuring the pitch diameter of threads.

We will only deal with the English measurement here. Should the operator have need for taking metric measurements with the metric system micrometer, it will be necessary for him to refer to instructions furnished with the micrometer or any good text book on machine shop practice.

The basis on which a micrometer operates is that the screw on the spindle, Fig. 47-C, has 40 threads to the inch. One complete turn of the spindle will then advance the spindle 1/40th, or .025 of an inch. Four turns will then advance

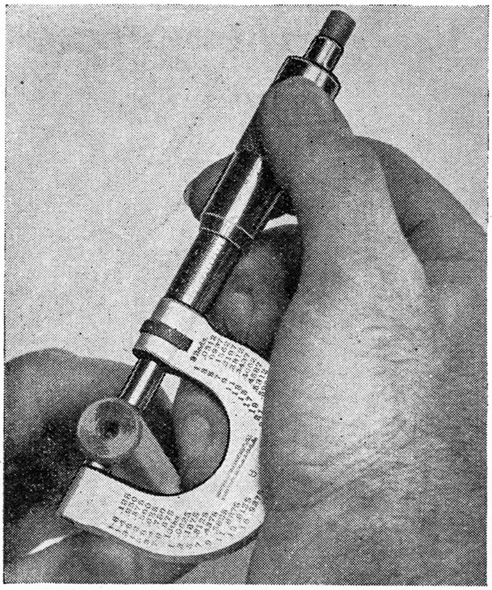


Fig. 49. "Getting" a micrometer measurement.

the spindle 4 times .025 or .100 or 1/10th of an inch. Each time the spindle is revolved one turn, a graduation appears on the barrel, Fig. 47-D. After each 4 graduations a number appears indicating each .100″ measurements.

The thimble, Fig. 47-E is graduated in 25 divisions and each one represents .001″ movement of the spindle. Every 5th graduation is numbered and facilitates the reading to the thousandth of an inch.

After these few simple rules are memorized by the operator, he need only practice taking sample readings. They should be made as follows: Be sure to use the same "feel" as was necessary when taking measurements with inside and outside calipers. Never force them over the work. Hold the "mike" between the thumb and forefinger as shown in Fig. 49, and let the weight of the micrometer "get" the diameter. Also, do not use the micrometer on a revolving piece as it will seriously impair its accuracy. Micrometers should be checked occasionally with a reference disk or standard gage. If they are out of adjustment, this should be corrected as soon as possible. A drop or two of fine oil should be placed on the screw occasionally. The micrometer should be kept away from abrasive particles and should not be dropped. These things will also result in inaccurate measurement. When the proper "feel" is obtained, the reading is then taken by observing the number of graduations visible on the barrel and spindle and reading them. For example, in Fig. 47, there are 7 visible graduations on the barrel, that will indicate 7 times .025″ or .175 inches. Add to this the number of graduations on the thimble. This reading is .003. The correct reading then is the sum of these two figures or .175 inches plus .003 inches equals .178 inches. The reading is then .178 inches.

Inside micrometers, Fig. 48, are used in a similar manner.

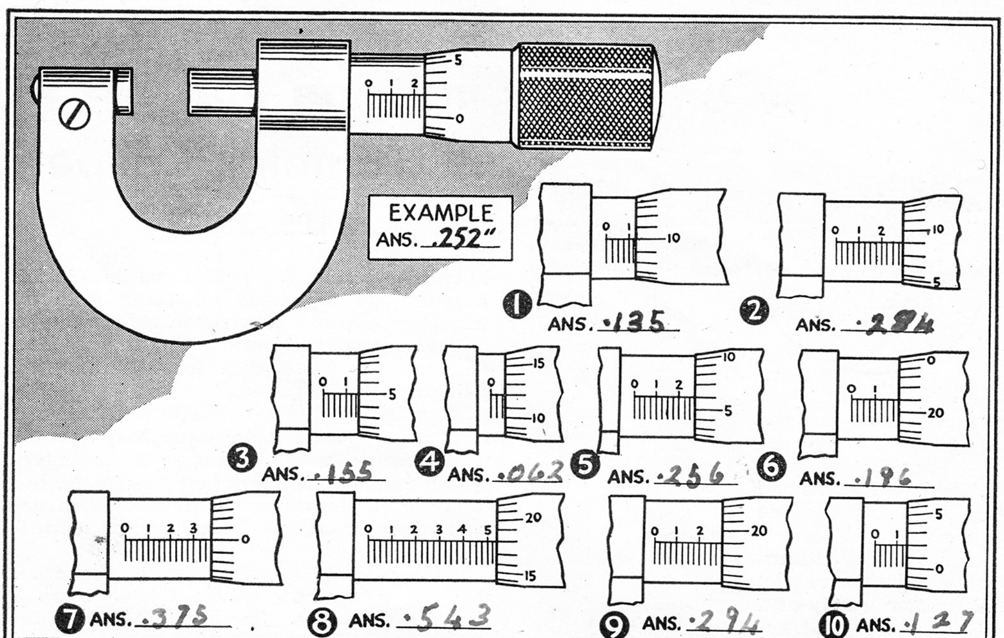


Fig. 50. Write in your answers to these settings, and see page 26.

Skill with Vernier Caliper also Necessary

LESSON 11

THE vernier caliper, Fig. 51, is a precision instrument used by the machinist. Reading and using this tool is of great importance. Like the caliper and micrometer it depends to a great extent upon proper "feel" for accurate results. The system of graduating vernier calipers is usually 24 divisions on the main scale and 25 divisions on the Vernier scale. Some vernier calipers have 19 on the main scale and 20 on the vernier scale. In both cases the difference between graduations is .001 of an inch. On the first system the main scale is divided into inch graduations and each inch divided into 40 equal spaces. Each space then equals 1/40th of an inch or .025″.

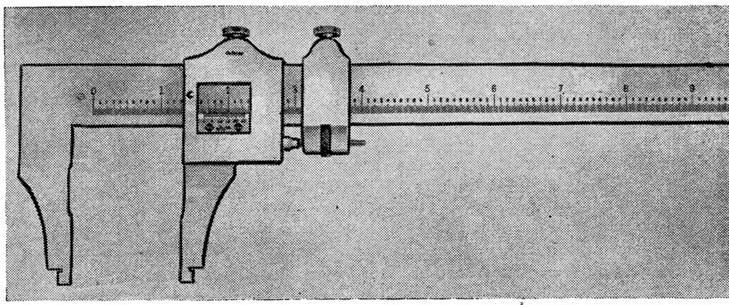


Fig. 51. Outside-inside vernier caliper. Tips of jaws used for inside measurements.

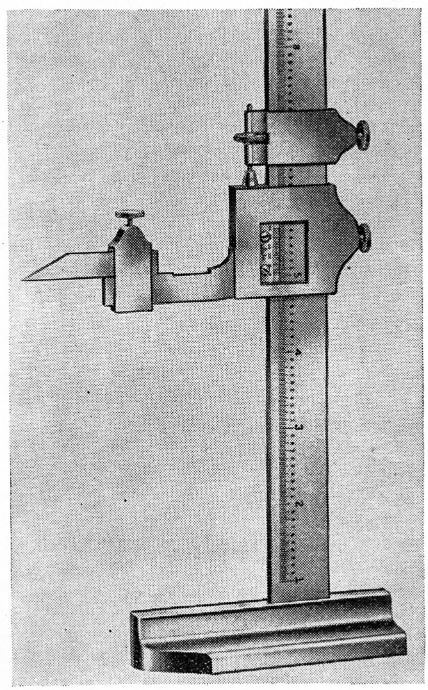


Fig. 52. Vernier height gage.

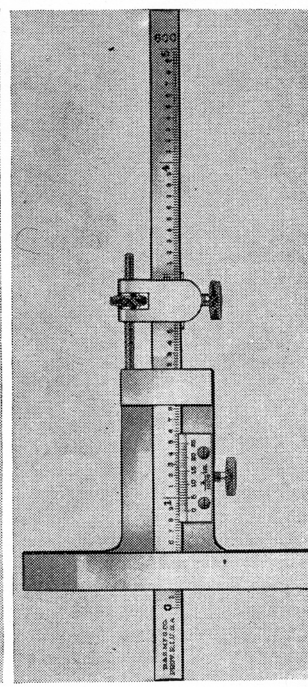


Fig. 53. Vernier depth gage.

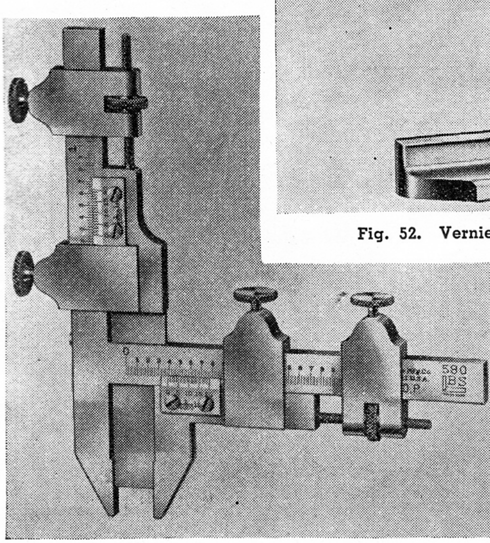


Fig. 54. Gear-tooth vernier caliper.

To read the vernier, read the main scale up to the zero mark on the plate, then look along the vernier scale until a line is found that lines up exactly with any line on the main scale. This reading, added to the reading nearest the zero mark on the vernier scale, will give the complete reading. For example: if the zero mark on the main scale was slightly over the 6 mark, that would indicate .600″ and if the 15 on the vernier scale lined up with a line on the main scale that would indicate 15 additional thousandths. This added to the original .600″ would equal .615″ or the final reading.

Other applications of the vernier besides in calipers are the vernier height gage, Fig. 52, that is used in layout work and inspections,

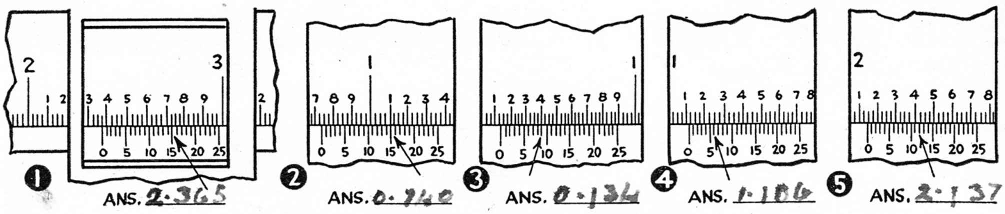


Fig. 55. Write in your answers to these settings, and see page 26.

vernier depth gage, Fig. 53, used similar to a depth micrometer, and gear tooth vernier caliper, Fig. 54, used for taking the cordal thickness of gear teeth, cutters, and hobs. Fig. 55 is a test sheet for the operator to try his skill in reading vernier measurements. The vernier principle may be applied to the micrometer barrel and is read in exactly the same manner.

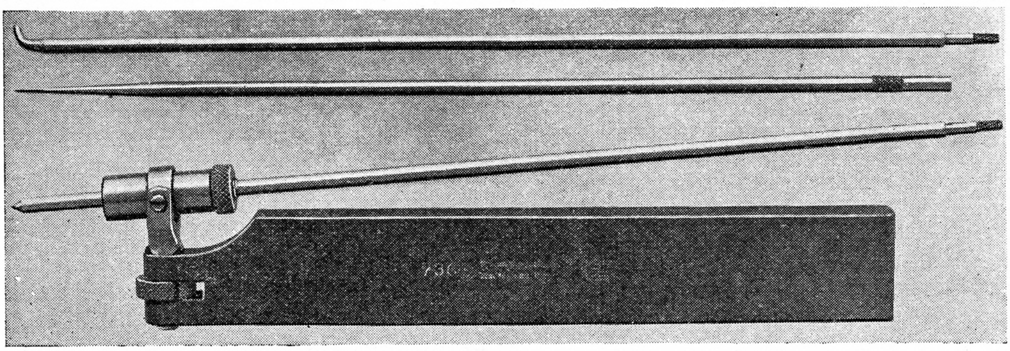


Fig. 56. "Wiggler" indicator for centering work.

Indicators for Setting and Testing Work

LESSON 12

INDICATORS are used to quite an extent by the lathe operator in accurate setting up and testing work on the lathe. Indicators are of various types, the "wiggler," Fig. 56, and the dial test indicator, Fig. 57, are examples.

A typical application of the wiggler might be in the testing and trueing of a job that is clamped to the face plate or held in a four-jaw chuck and has but a center punch mark to aid in locating it in its proper position. One end of the testing bar is placed in this mark and when the spindle is rotated the other end of the wiggler should run true if the piece is accurately located. Because the pivot is closer to the work end, the outer end exaggerates the amount the center point may be off, thus facilitating more accurate location by the operator.

The universal dial test indicator, Fig. 57, has many applications on the lathe, and it is usually held or clamped in the tool post. The dial is graduated in thousandths of an inch, and is so arranged that a small button on the bottom may be adjusted to rest on the work being tested. This type of indicator is very sensitive and practical. A typical example of its use is in the alignment of the lathe centers, and this will be discussed in another lesson. Another use for this type of indicator is in chucking up work of cylindrical shape in an independent four-jaw chuck. The lathe operator can readily find additional application for these tools.

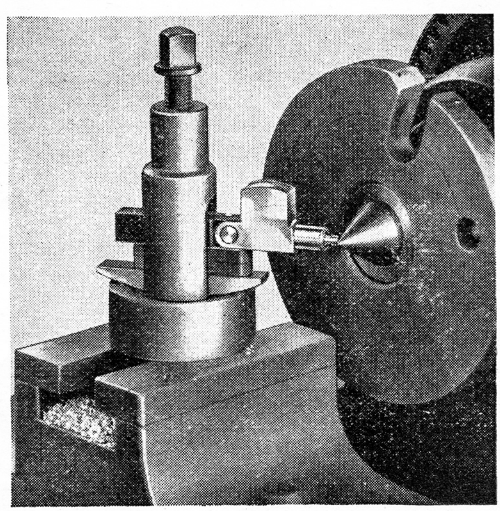


Fig. 57. Dial test indicator.

Working with Drills and Reamers

LESSON 13

DRILLING and reaming in lathe work may be performed in manners similar to those on a drill press using like equipment. The drill or reamer may be held either in the headstock spindle and caused to rotate, or held in a drill chuck, or in case it has a tapered shank it is sleeved up to fit the tailstock spindle and the work caused to rotate.

The lathe operator should have a good understanding of the terminology with reference to drills and reamer parts, clearances, speeds, and feeds. He should acquaint himself with the correct terminology of the drill parts as shown in Figs. 58-A and 58-B. While drills vary as to types, their cutting action is very much the same. A drill may be defined as a cylindrical piece of tool steel having two equal and diametrically opposite spiral grooves or flutes that form cutting edges on the cone-shaped end. The cutting takes place at this cone-shaped end and the flutes permit escape

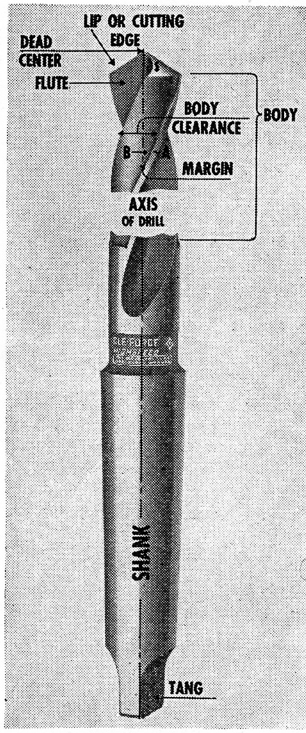


Fig. 58-A. Parts of a twist drill.

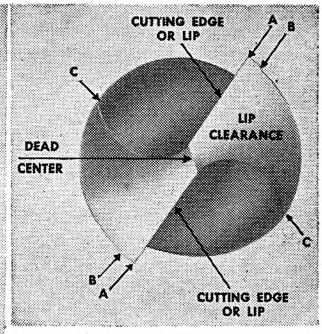


Fig. 58-B. Point of a twist drill.

Fig. 59. Chucking reamer.

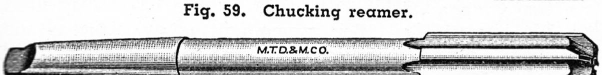


Fig. 60. Rose reamer.

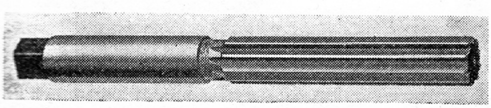


Fig. 61. Solid reamer.

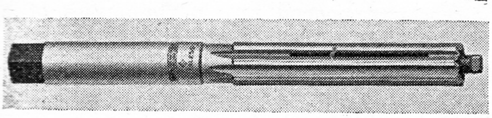


Fig. 62. Adjustable reamer.

of the material that is removed from the work. These flutes also permit the lubricant or coolant, when one is used, to get to the cutting point.

There are four classifications of drills as to sizes. The one most frequently used is the fractional size drill. These can be secured usually from $1/16''$ to $1''$ in $1/64''$ graded sizes. Above this size, the variation between sizes is sometimes slightly more.

Another system of drill classification is the number size. These range from No. 1 to No. 80 as standard, the No. 1 being the largest and No. 80 the smallest. These are of decimal measurements, and it is necessary to refer to a chart or gage for the size of each number. They are very commonly used by the mechanic in drilling tap size holes when using numbered taps.

The third system is the alphabet or letter size drills which run from A to Z inclusive. A is the smallest size which is slightly larger than the No. 1 drill. The lettered drills carry on from there in decimal sizes up to the Z drill which is .413″.

Still another system is the millimeter sized drills. The millimeter classification is very seldom used but many times use is found for them for in-between sizes that are not standard with the other systems. They are graduated in millimeter and half-millimeter variations. These can quickly be transferred into decimal equivalents through the use of a conversion chart.

The speed at which drills should operate is similar to that for turning operations and may be calculated by using the same formula, or any handbook may be referred to for this information. It is very important to have this information in order to work up to capacity. When drilling in the lathe, it is usually necessary to spot the hole first either with the special turning tool or by the use of a short stub of a drill, such as a combination center drill. This is necessary to prevent the drill from drilling off center. A small drill used after the spotting operation will

tend to keep the drill in line and will relieve some of the pressure necessary when using large drills. However, the drill usually drills a hole slightly off center in spite of all this, and it is customary practice if the hole must run true to follow the drilling operation by boring. In this way there will be absolute concentricity.

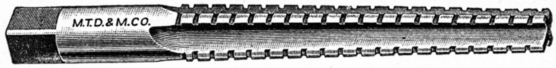


Fig. 63. Roughing reamer.

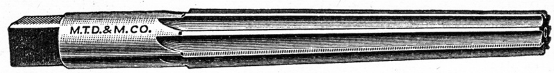


Fig. 64. Finishing reamer.

Reamers are cutting tools similar to the drills except that the flutes are many and run the length of the body, either straight or spiraled. They are used for giving an accurate size and finish to a previously drilled or bored hole. A very small amount of stock should be left for this reaming operation. Usually .005″ to .010″ is sufficient for holes up to 1″.

There are many types of reamers, some are used by machine, such as the chucking, Fig. 59, and rose reamers, Fig. 60. There are other kinds which are more or less special which have expanding or adjustable features that are used on production. The hand reamers, Fig. 61, may be solid or adjustable, Fig. 62, and are used only at the bench. The operator should be very careful and never turn a reamer backwards, as this practice will destroy the accuracy of the reamer.

The cutting edges of all reamers, like other edge cutting tools, should be protected when not in use and should be checked for size by taking micrometer measurements before using. Reamers may be also had in taper form for both roughing, Fig. 63, and finishing, Fig. 64, of the various kinds of tapers. The operation of grinding drills and using them and reamers will be discussed in future lessons.

Things to Know About Tapers

LESSON 14

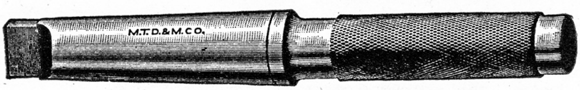


Fig. 65. Taper plug gage.

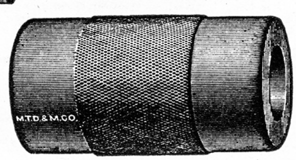


Fig. 66. Taper ring gage.

THE machinist should be well acquainted with the various systems of tapers as used in machine construction. The taper may be defined as a uniform gradual increase or decrease in size from one end to the other on either cylindrical or flat surfaces. We will deal here with the cylindrical shapes only. The taper on the part is usually expressed in the amount it tapers per inch or per foot. Most all machines depend on tapers for holding tools. This is a very good method because it is quick and sure as it lines up the tools accurately and holds them rigidly, providing the operator keeps the shanks clean and free from nicks and burrs. The mating part must likewise be kept clean.

There are three standard systems of tapers in common use. They are: 1. The *Morse* taper which is used in most drill presses and on tapered shank drills. It varies in size and has a taper per foot of .600″ to .630″ taper per foot. 2. *Brown and Sharpe* taper which is used in most milling machines has a taper of .500″ on all sizes up to No. 10 where it changes to .516″ taper per foot. 3. The *Jarno* taper is the most standard of all three but it is the least used because of its recent development. It has a constant taper on all sizes of .600″ taper per foot. It is used by some lathe manufacturers for head and tailstock spindles. There are still other tapers such as used on taper pins and for special application. These will not be dealt with here.

As taper systems are standard, one can quickly get the information regarding any size or make by referring to a handbook.

Tapers may be produced on the lathe by a number of methods. The one most commonly used for slight tapers is the *taper attachment method.* One frequently used when no taper attachment is available is the *tailstock offset method,* but it can be used only for external tapers with a limited amount of taper.

For steep tapers the compound rest is set at an angle to accomplish the desired results. Tapers may also be formed by the use of a broad nose form tool or by faking with a narrow tool when extreme accuracy is not important. Internal tapers may be produced by either using the attachment, compound rest, or by means of a taper reamer.

The method of performing taper operations will not be discussed at this time, as a lesson on this subject is scheduled later in this course.

The operator should know how to check tapers. Tapers are usually inspected by means of taper ring gages, Fig. 66, for external and taper plug gages, Fig. 65, for internal tapers. When using these instruments for checking, the degree of accuracy is indicated very quickly. If it is correct, no movement of the gage will be possible in a sidewise direction. The operator should avoid a twisting motion when using these gages.

A method must be devised for checking tappers that are not standard. This method usually involves trigonometric calculation and the operator may find it necessary to consult a handbook for aid in their solution.

External tapers may be checked with a common micrometer if the operator uses care and precision. The best method is to cut a sample taper on scrap material first. If he knows what the taper per foot is, this can be divided into the part that he is to measure. For example: Take a standard Brown & Sharpe taper No. 7, the taper per foot is .600″. If he turns for 2″, the piece should taper ⅙ of .600″ or .100″ and the micrometer readings taken 2″ apart on the tapered surface should indicate this amount. If the taper indicated is incorrect, adjustments should be made to correct this error. When this taper is found to be accurate, the operator may proceed to bring the piece to size.

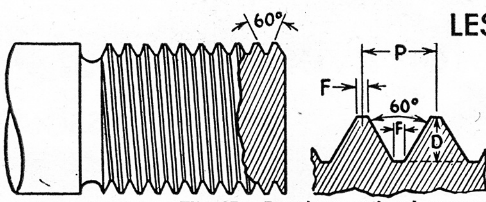
When cutting tapers on the lathe, it is very important that the tool bit be set exactly center high, and when using the taper attachment, it is equally important that the compound rest be set to feed straight in. This simplifies the method so that the operator may remove the correct amount of material to bring the taper to size without spoiling a piece of work.

A simple example is as follows: If the measurements indicate that a taper ring gage must fit ⅝″ from the end of the shaft and it now extends over the end ¼″, the operator must remove an unknown amount of material to accomplish this result. If the operator takes the amount of taper per inch and breaks it down as follows, he can quickly set the compound rest in a desired amount: Supposing we have the above taper of .600″ per foot, then the taper per inch is .050″. On most compound rests, dials are graduated in thousandths of an inch and in moving it in .001″ it removes two-thousandths of material, the following should be true. By turning straight in .025″ a taper ring gage would slide on 1″ farther. If the compound rest is turned in .012½″ the gage would slip on ½″. If turned in .006¼″, the taper slides on ¼″, and if turned in .003⅛″ the taper slides on ⅛″. By this method the operator can quickly calculate the necessary amount to advance the gage the required ⅞″ as indicated is necessary in the above example, or ½″ equals .012½″ plus ¼″ or .006¼″ infeed plus ⅛″ or .003⅛″, thereby totaling ⅞″ travel with an infeed of .021⅞″. This is really not as complicated as it sounds. All the operator need do is to try this method once, and it becomes a fixed practice with him.

The various methods of setting up the engine lathe and how to perform these taper, turning and boring operations will be discussed in detail in future lessons covering the subject.

Learn Thread Terminology Too

LESSON 15



FORMULA

$P = \text{Pitch} = \dfrac{1}{\text{No. Th'ds. Per In.}}$

$D = \text{Depth} = \text{P. x } .64952$

$F = \text{Flat} = \dfrac{P}{8}$

Fig. 67. American national screw thread form.

BEFORE cutting threads on a lathe, the operator should have an understanding of the various parts of the thread and the various kinds of threads and their uses. Screw threads are considered more or less standard and the *American National Series* now replaces the old U. S. Standard and the S. A. E. system. Fig. 67 shows this system of thread. This is one that is most commonly used on most nuts, bolts, and special fastenings.

There are other systems of threads that are used for special applications as follows: The *Acme thread,* Fig. 68, is used for most adjusting screws on machines. The *Brown & Sharpe* 29-degree worm thread, Fig. 69, while similar to the Acme standard, is used on worm gears. The *square thread,* Fig. 70, is a thread that is used mostly in screw jacks. Another thread which is considered standard in England is the *Whitworth* thread, Fig. 71. Still another system, while very seldom used, is the *buttress* thread as shown in Fig. 72. It is primarily used on vise screws where the thrust is only in one direction. *Multiple threads,* Fig. 73, of any of these classifica-

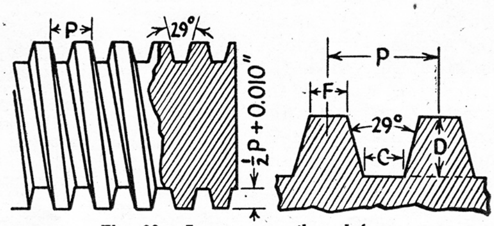


Fig. 68. Acme screw thread form.

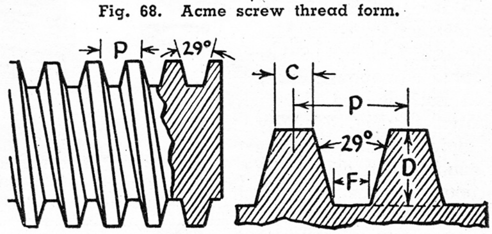
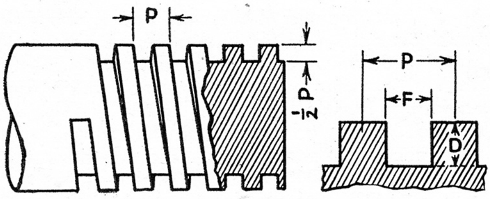


Fig. 69. 29 Degree worm thread form.



FORMULA

$$P = \text{Pitch} = \frac{1}{\text{No. Th'ds. Per In.}}$$

$$D = \text{Depth} = P. \times .500$$

$$F = \text{Space} = P. \times .500$$

Fig. 70. Design and proportions of square screw thread.

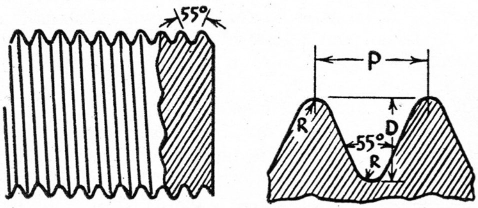


Fig. 71. Whitworth screw thread form.

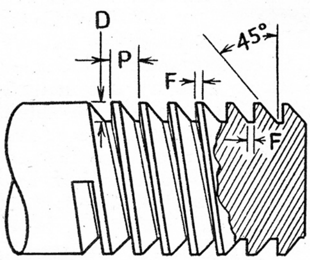


Fig. 72. Buttress thread form.

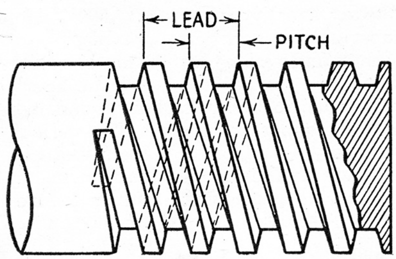


Fig. 73. Multiple screw thread having two grooves (double thread).

tions may be produced on the lathe. A multiple thread consists of two or more grooves which causes more rapid advancement or movement of the part in conjunction with it. It is necessary in producing this type of thread to have an equivalent number of equally spaced slots in the driving plate as threads desired on the work. There are other methods, but this one is the most used.

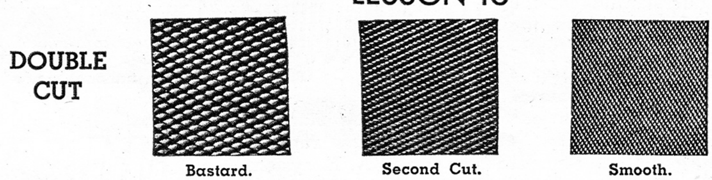
Threads may also be produced on the lathe through the use of taps and dies, but while this method is not quite as accurate, it serves its purpose on production of parts not requiring precise measurement.

Detailed information with reference to exact measurements for the various size threads may be secured from any handbook. The actual thread cutting operation will be discussed in a later lesson. The thread cutting operation is one of the most interesting to perform and the operator should take pride in being able to do a good job and increasing his skill.

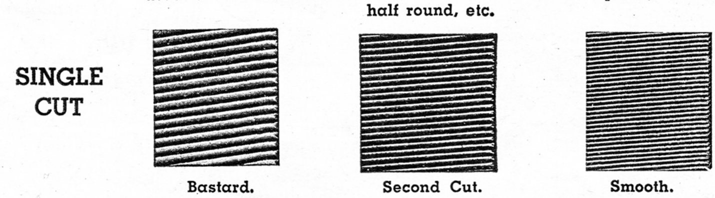
Files for Every Kind of Job

LESSON 16

FOR general machine shop work, machinist's files, Fig. 74, are of three degrees of coarseness or tooth spacing: (1) bastard, which is the coarsest, (2) second-cut which is medium, and (3) smooth-cut which is the finest. Bastard-cut files are for removing metal rapidly; second-cut and smooth-cut files are for leveling off or finishing and for use on all soft or tempered metals. Machinist's files are either double cut or single cut and include the shapes shown in Fig. 75. By studying this chart, the mechanic can select the proper file for the use desired. Take good care of your files.



Double Cut file—has two courses of teeth or chisel cuts crossing each other, one course being finer than the other. Double Cut is used on all machinists' files, such as flat, hand, square, round, half round, etc.



Single Cut file—has one unbroken course of teeth or chisel cuts across its surface, parallel with each other, but at an oblique angle to the length of the file. The single cut is used on mill files, on the taper files and on special types of some saw files.

Fig. 74. Standard cuts of file teeth.

SHAPE	CROSS SECTION	WIDTH OR FACE from heel to middle	WIDTH OR FACE from middle to point	THICKNESS OR EDGE from heel to middle	THICKNESS OR EDGE from middle to point	GENERAL USES
Flat		Uniform	Tapered	Uniform	Tapered	A general-purpose file. Good for flat surfaces.
Hand		Uniform	Uniform	Uniform	Tapered	Another general-purpose file for angles, corners, and flat surfaces.
Pillar		Uniform	Uniform	Uniform	Tapered	Keyways, slots and narrow work.
Warding		Tapered	Tapered	Uniform	Uniform	Narrow work requiring thin file. Making keys.
Square		Uniform from heel to middle; then tapered to point				Corners, grooves, keyways, slots.
Three Square		Uniform from heel to middle; then tapered to point				Acute angles, corners, grooves, notches.
Round		Uniform from heel to middle; then tapered to point				Holes, shaping curved surfaces.
Half Round		Uniform from heel to middle; then tapered to point				Concave corners, crevices, rounding holes.
Knife		Uniform from heel to middle; then tapered to point				Cleaning out acute angles, corners, slots.

Fig. 75. File shapes and their uses.

Taps and Tap Drill Sizes

LESSON 17

TAPS produce internal threads at the bench or may be used on tapping machines and sometimes, with extreme care and caution on lathes. Taps usually come in sets of three, Fig. 76, taper, plug, and bottoming. These taps are all full size and any one may be used without the others. The most commonly used one is the plug tap and the least used is the bottoming tap. The bottoming tap is necessary only when threads must be all the way to the bottom of a hole. As a feature of design this is considered bad practice as it consumes considerable time to accomplish and is very seldom necessary.

The operator should do everything in his power to prevent the breaking of taps. Should this happen, as it frequently does, it is necessary to remove the broken tap by one of several methods.

Fig. 77. Tap extractor.

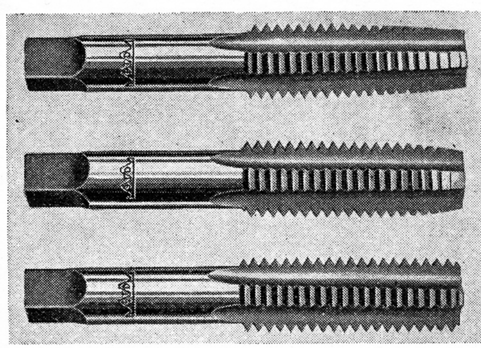


Fig. 76. Taper, plug, and bottoming taps, reading from top down. Taps come in sets of three.

Tap extractors, Fig. 77, may be purchased for this purpose. However, if the tap is firmly imbedded in the work, the operator may break the fingers of the extractor and be worse off.

Sometimes a tap of the larger sizes may be removed by the operator tapping alternately on opposite sides of a broken tap or better still, two persons tapping in unison will work better.

Another method is to apply a few drops of acid to help eat away the material and cause easier removal of the broken tap. The action of the acid should be stopped by washing with water as soon as possible to avoid removal of too much material.

Still another method is to anneal the whole

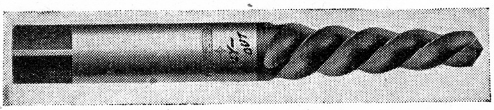


Fig. 78. Screw extractor.

piece in the annealing furnace or by localized annealing with a gas torch, then drill a hole in the tap and remove with a tapered tang of a file or even with a screw extractor, Fig. 78.

The chart, Fig. 79, will give the operator the basic tap drill sizes for the American National thread series. Should the operator find it necessary to cut specials, this information may be had from handbooks. A simple formula for determining tap drill sizes is as follows: The tap drill to produce a 75% thread is: T equals the major diameter D minus the constant .97 divided by the number of threads per inch N. $T = D - (.97 \div N)$. When this figure is obtained it will be necessary for the operator to look on a table of drill sizes to find the nearest size drill. When tapping hard material it is best to use a slightly larger drill.

TAP DRILLS
AMERICAN NATIONAL
COARSE AND FINE THREADS

THREAD NOMINAL SIZE	DRILL SIZE	DRILL DECIMAL	THREAD NOMINAL SIZE	DRILL SIZE	DRILL DECIMAL
0-80	3/64	.047	5/16-18	F	.257
1-64	53	.060	24	I	.272
72	53	.060	3/8-16	5/16	.313
2-56	50	.070	24	Q	.332
64	50	.070	7/16-14	U	.368
3-48	47	.079	20	25/64	.391
56	45	.082	1/2-13	27/64	.422
4-40	43	.089	20	29/64	.453
48	42	.094	9/16-12	31/64	.484
5-40	38	.102	18	33/64	.516
44	37	.104	5/8-11	17/32	.531
6-32	36	.107	18	37/64	.578
40	33	.113	3/4-10	21/32	656
8-32	29	.136	16	11/16	.688
36	29	.136	7/8- 9	49/64	.766
10-24	25	.150	14	13/16	.813
32	21	.159	1 - 8	7/8	.875
12-24	16	.177	14	15/16	.938
28	14	.182	1 1/8- 7	63/64	.984
1/4-20	7	.201	12	1 3/64	1.047
28	3	.213	1 1/4- 7	1 7/64	1.109
			12	1 11/64	1.172

TAP DRILLS
AMERICAN NATIONAL
COARSE AND FINE THREADS

THREAD NOMINAL SIZE	DRILL SIZE	DRILL DECIMAL	THREAD NOMINAL SIZE	DRILL SIZE	DRILL DECIMAL
1 3/8- 6	1 7/32	1.219	2 1/2- 4	2 1/4	2.250
12	1 19/64	1.297	2 3/4- 4	2 1/2	2.500
1 1/2- 6	1 11/32	1.344	3 - 4	2 3/4	2.750
12	1 27/64	1.422	3 1/4- 4	3	3.000
1 3/4- 5	1 9/16	1.563	3 1/2- 4	3 1/4	3.250
2 - 4 1/2	1 25/32	1.781	3 3/4- 4	3 1/2	3.500
2 1/4- 4 1/2	2 1/32	2.031	4 - 4	3 3/4	3.750

AMERICAN NATIONAL PIPE THREADS

THREAD NOMINAL SIZE	DRILL SIZE	DRILL DECIMAL	THREAD NOMINAL SIZE	DRILL SIZE	DRILL DECIMAL
1/8-27	R	.339	1 1/2-11 1/2	1 47/64	1.734
1/4-18	7/16	.437	2 -11 1/2	2 7/32	2.218
3/8-18	37/64	.578	2 1/2- 8	2 5/8	2.625
1/2-14	23/32	.719	3 - 8	3 1/4	3.250
3/4-14	59/64	.921	3 1/2- 8	3 3/4	3.750
1 -11 1/2	1 5/32	1.156	4 - 8	4 1/4	4.250
1 1/4-11 1/2	1 1/2	1.500			

Fig. 79. Basic tap drill sizes.

Cutting Lubricants Dissipate Heat

LESSON 18

CUTTING lubricants are very seldom used when only a few parts are being turned out, and in some cases cutting lubricants are never used. To speed up production, lubricants are used to dissipate the heat. The chart, Fig. 80, gives the operator the recommended coolants for use in the majority of cases.

LUBRICANTS FOR CUTTING TOOLS

Operation	Cast Iron	Machine Steel	Copper	Brass, Bronze	Aluminum	Lead Babbitt
Turning or Boring	Dry	Dry-oil, soda-water	Milk	Dry	Kerosene	Dry
Cutting off or Grooving	Dry	Oil or soda-water	Milk	Dry	Kerosene or Turpentine	Dry
Screw Cutting	Dry	Oil	Milk	Dry	Kerosene or Turpentine	Dry
Threading with dies	Dry	Oil	Milk	Dry	Kerosene or Turpentine	Oil
Tapping	Dry	Oil	Milk	Oil	Kerosene or Turpentine	Oil
Drilling, Counter-boring	Dry	Oil or soda-water	Milk	Dry	Kerosene or Turpentine	Oil
Knurling	Oil	Oil	Milk	Oil	Kerosene or Turpentine	Dry
Reaming	Dry	Oil	Milk	Dry	Kerosene or Turpentine	Dry
Filing	Dry	Oil or dry	Milk or dry	Dry	Kerosene or Turpentine	Oil
Polishing with emery cloth	Oil	Oil	Oil	Oil	Kerosene or Turpentine	Oil

Fig. 80. Lubricants recommended for cutting tools.

Basic Facts About Heat Treating

LESSON 19

AN UNDERSTANDING of the terminology of heat treating operations and what takes place is important to the beginner of shop practice. Heat treating may be defined as the changing of the physical properties of the metal by the proper application of heat. The amount of carbon content with reference to steel is the determining factor of its heat-treating properties. While other materials than steel may be heat treated, we will deal more or less specifically with this one phase.

Hardening of steel is done by heating it slightly above its critical point. This critical point may be found by referring to a chart. However, another method used when this is not known, is to rely upon the magnetic test. Steel loses its power to attract a magnet when it has reached this point. It is then quenched in the proper quenching medium, usually water, brine or oil. It is then necessary because of its glass hardness, to draw this temper to the right degree of toughness for a job the tool is called upon to perform.

Tempering, therefore, is the process of reheating a piece of metal after hardening for the purpose of reducing the hardness or removing its brittleness. Any handbook will give the proper temperature or oxidizing colors necessary for the purpose the tool is to serve.

Quenching may then be described as the rapid cooling of a heated piece of material in water, oil, still air, and sometimes in a blast of air.

Annealing is the softening or removing of hardness which refines the grain structure and makes the piece ductile or machinable.

Normalizing is performed on the metal by heating it to approximately 100 degrees above the critical point and allowing it to cool off gradually. This will restore the steel to its normal physical condition and any strains set up by machining will have been removed.

Carburizing may be defined as the process of increasing carbon content on mild steel at the surface to a predetermined depth, thereby, causing this surface to become tool steel so that it may be hardened. This is accomplished by heating it in an atmosphere of carbonaceous material or gas. Some materials used for this purpose are leather scraps, charcoal, bone dust, charred peach stones and commercial carburizing pellets.

Cyaniding, while similar to carburizing, is slightly different. The mild steel is heated in contact with molten cyanide and then quenched. The thickness of the case seldom exceeds .010″.

Typical examples of heat treatments for steels used in the shop are as follows: Mild steel, commonly called cold-rolled steel, with an average reading of S A E 1020 may be heat treated by cyaniding or carburizing. Tool steel of the S A E 1095 variety is heated 1425° to 1475° F. quenched in brine and tempered from 250° to 600° F. depending on the purpose of the tool. High speed steel is first preheated to 1400° or 1500° F. It is then placed in a super heat around 2300° to 2400° F. and quenched in oil. Some protection is necessary at this high heat to prevent scaling of the surface. The piece is then tempered around 1000° or 1100° F. to bring it to the proper hardness.

Hardness of metal may be checked with a fine file. This method, however, is very inaccurate except in the hands of an experienced heat treater. There are more common methods of testing hardness with the aid of such instruments as the Rockwell, Brinell, and Scleroscope testing machines. These will not be discussed now because just a brief explanation to the user is all that is necessary to perform operations, and this will be made in later lessons.

Making an Operation Sheet

LESSON 20

THE operator of a lathe or any machinist, for that matter, should be able to make up an operation sheet on how to produce a piece of work. Before starting, the first thing the operator should do is to study the print in order to know what is expected as the final product. When he is quite sure of himself, he may proceed in an outlined form to jot down such information as called for in the chart, Fig. 81.

Other things which bear watching are the inspection of the raw material for defects and to see if there is sufficient material for removal to bring it to size.

After the operator has studied these lessons and is quite certain that he knows this material or where to find the answer to most of his questions, he is then ready to take on additional responsibilities. These few lessons in no way cover all of the information that a lathe operator is required to know. It is, however, a start in that direction. In order to advance himself, the machine operator should continue his studies by going to night school, studying by correspondence and by keeping abreast of current literature in his field.

SCHEDULE OF OPERATIONS

Name of Job__________ Foreman's O.K. (__________)

Size of Stock__________ Job No.__________

Operator__________ Kind of Material__________

Date__________

No.	Operation	Est. Time Min.	Actual Time Min.	Machine and Tools

Fig. 81. Every lathe operator is expected to know how to make out a schedule of operations.

The lathe operator may well profit by keeping an operation schedule on file for each job that he performs. Such information as speed and feed settings also have a place under this heading as do special notes on stop settings, jigs, fixtures, and unusual conditions that must be remedied.

In this library of information, the conscientious operator should keep news articles on his trade and manufacturers' bulletins on his machine or material with which he is working. Most companies obligingly send this information to the shop man upon request.

Answers to Micrometer and Vernier Settings

MICROMETER SETTINGS ON PAGE 16
(1) .135 (2) .284 (3) .155 (4) .062 (5) .256
(6) .196 (7) .375 (8) .543 (9) .294 (10) .127

VERNIER SETTINGS ON PAGE 18
(1) 2.365 (2) .740 (3) .134 (4) 1.106 (5) 2.137

Always Begin by Aligning Centers

LESSON 21

NOW that the lathe operator has become familiar with the lathe, its parts and their function, he is ready to study the individual operations that can be performed on it. These operations will be taken up one by one in this lesson and those to follow. Ease in learning and the frequency of use, as far as possible, have governed the order in which the operations are here presented. The student may vary the sequence as determined by the type of work or product turned out by the shop in which he is working.

The first thing the operator of any lathe should do each and every time he prepares the lathe for a machining operation is to check and align the centers. The accuracy of the work done on a part is dependent upon these points.

The live center should first be removed from the spindle, cleaned and replaced in the same position each time. This can be facilitated by placing witness marks on the spindle and center and matching these marks each time. In this position, once the center is machined to a 60 degree angle as shown in Figs. 82 and 83, it should run true and trueing it in this manner will not be necessary each time the lathe is used for turning work between centers. The good operator should start the machine revolving and observe whether the point of the live center runs true. Visual examination will note any large error in this respect but should the work require absolute concentricity the center may be tested with the dial indicator as shown in Fig. 84. If there is

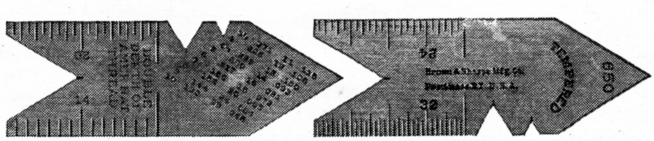


Fig. 82. Back and front sides (left to right) of center gage for testing the angle of a lathe center point.

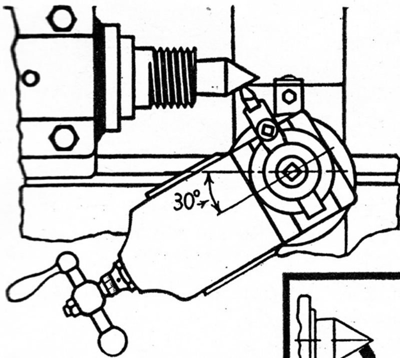


Fig. 83. Machining a 60 degree lathe center point.

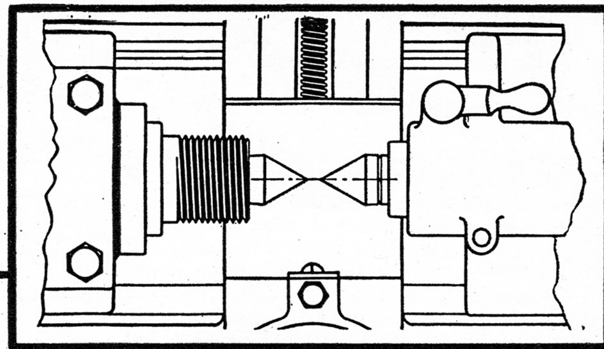


Fig. 85. Checking the alignment oı a lathe center.

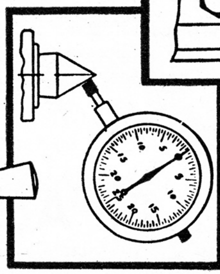


Fig. 84. Testing the center with a dial indicator.

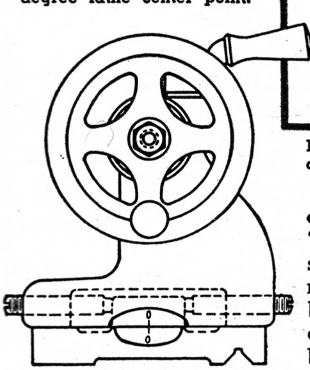


Fig. 86. Tailstock set on center by adjusting screws at each side.

even the slightest "run out," it will show up. This must be corrected by turning if the center is soft or by grinding if a hardened center is used. With careful handling this live center will run axially true. The main causes for run out are chips or dirt in the spindle or on the center itself, or bending of the point by dropping the center while removing it, or allowing the lathe dog to hit the tool holder or compound rest while turning. Any or all of these can be easily avoided by the careful operator, but should they accidentally happen, the operator must know how to put the center back into shape again.

When the operator is reasonably sure of the live center running true, his next job is to check the tailstock or dead center for approximate alignment with the turning axis of the lathe. Approximate alignment may be checked in either of the following ways: First, as shown in Fig. 85, bring the tailstock center up gradually until it is a short distance from the live center and by careful visual examination the slightest amount of out of alignment can be readily noticed. A piece of white paper placed directly below this point will accentuate the error. If the centers are off, adjustment is made by adjusting the screws in the tailstock shown in Fig. 86. The second method of checking the centers for approximate alignment is by noting the position of the index marks on the end of the tailstock, Fig. 86, and adjusting accordingly. The first method is preferred by most operators.

Each time the lathe is used for turning between centers it is also a good policy to remove this dead center, clean and replace it according to witness mark. This center has a 60 degree point to mate with the center holes. It is always hard as the work revolves on this point and the surface must be ground occasionally to keep it in good shape.

Should the work being turned require absolute accuracy of center alignment, it is necessary for the operator to use exact methods to obtain it. This may be accomplished by turning a scrap piece of work for the distance required and measuring at each end or even by using the project itself and turning two spots as shown in Fig. 87, using the same cross dial setting and measuring these places with a caliper or micrometer. If inaccuracy is present, it can be adjusted by turning the set screws in the tailstock a small amount and more trial cuts taken until both places measure identically. This is sometimes termed the trial and error method.

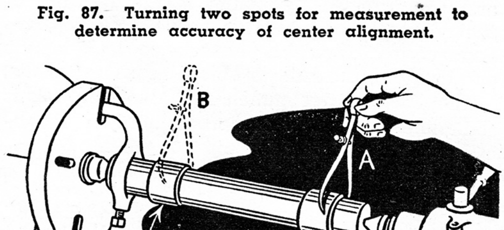


Fig. 87. Turning two spots for measurement to determine accuracy of center alignment.

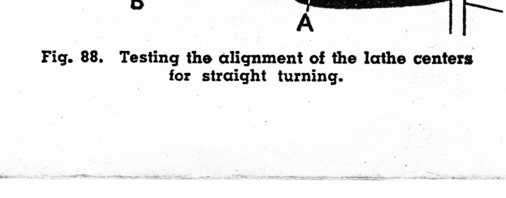


Fig. 88. Testing the alignment of the lathe centers for straight turning.

Fig. 88 is another way similar to the one above, but a piece of stock made especially for testing is used and the two collar portions have a cut taken off of them and measurements checked and adjustments made until readings are the same.

Sometimes the work does not lend itself to such trial and error methods, so the operator resorts to the use of a proof bar placed between centers and a dial indicator clamped in the tool post for checking the center alignment and adjusting it until there is no indication of out of alignment, Fig. 89.

When the above operations are completed the operator is reasonably sure of his machine and the quality of the work it will turn out. The time for performing this work is usually negligible when one considers that the piece might be scrap if after it is finished, inaccuracies are found to be present.

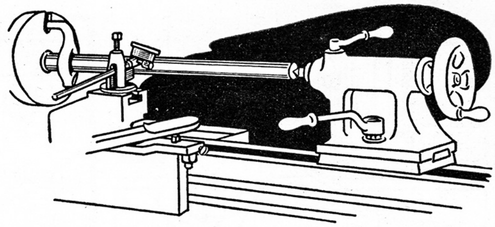


Fig. 89. Using a proof bar and dial indicator for checking centers alignment.

The operator should be always on the alert for dirt or chips on the center point and in the center holes as this, too, will result in "run out." A good operator observes these points.

Center Holes Must Be Accurate

LESSON 22

WORK to be done on the lathe must be provided with center holes of the correct size and shape, as was indicated in Lesson 4.

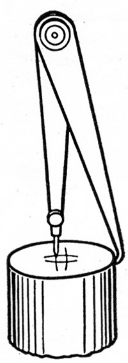
The approximate center of the stock must first be located. This may be accomplished in any of a number of ways.

The "Morphy method," Fig. 90. This is used most always on rough castings or out of round stock, but it can be used on perfectly round smooth stock with equal results. The hermaphrodite calipers are first set to a distance slightly greater than half the diameter and an arc scribed on the surface end of the stock which previously may or may not have been covered with layout fluid or chalk. The "morphy" calipers, or stock, is rotated a quarter of a turn and another arc scribed. This procedure is followed until four such arcs form a small square near the center of work.

The "center square method," Fig. 91 is commonly used on smooth round work and is quicker and more exact. It is self-explanatory as lines are scribed at right angles to each other and the intersecting points is the location of the center. The operator needs only a sharp scriber and draw his lines once and in the corner along the rule to obtain accurate results.

The "bell center punch method," Fig. 92, is sometimes used on small production but its accuracy is questioned by some mechanics, especially on rough, out of round stock or if the ends are not square.

The "surface gage method," Fig. 93, may be used on any even or odd shaped piece. The point of the scriber is set on approximate center made by scale measurement, a line is scribed and the work rotated 90 degrees and another line scribed.



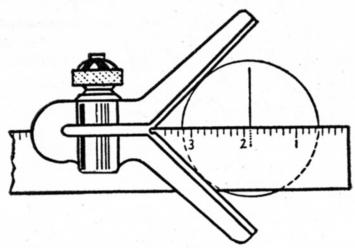


Fig. 91. Use of center head to locate centers, above.

Fig. 90. Centering with a hermaphrodite calipers.

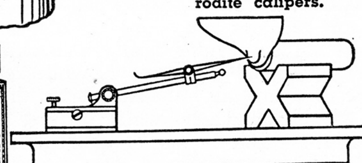


Fig. 93. Centering an irregular shape, above.

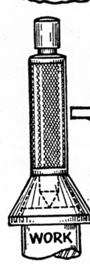


Fig. 92. A bell center punch.

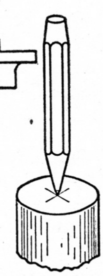


Fig. 94. Punching the center.

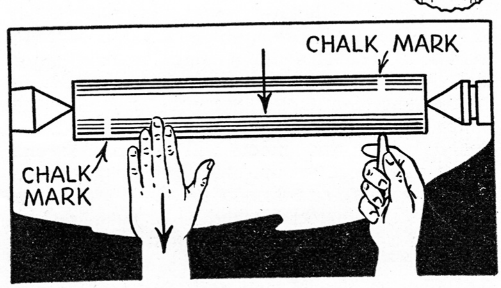


Fig. 95. Testing center punch marks.

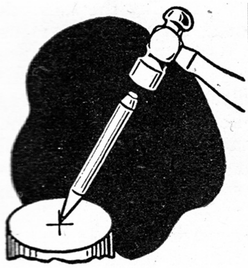


Fig. 96. Changing the location of a center punch mark.

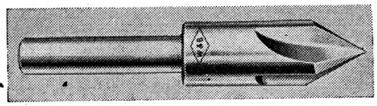


Fig. 97. A 60 degree countersink.

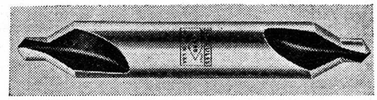


Fig. 98. Combination center drill and countersink.

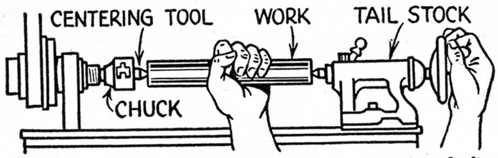


Fig. 99. Drilling the center hole in the end of a shaft.

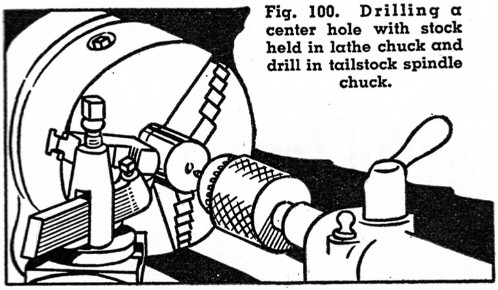


Fig. 100. Drilling a center hole with stock held in lathe chuck and drill in tailstock spindle chuck.

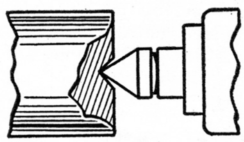


Fig. 101. A poorly drilled center hole, too shallow and incorrect angle.

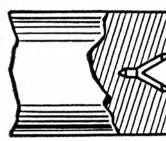


Fig. 102. An incorrect center hole, drilled too deep to fit center.

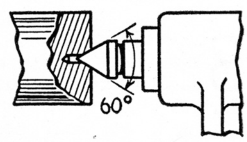


Fig. 103. A correctly drilled and countersunk hole fits the center.

The methods described above are used when only one or a few parts need centering; other methods will be described later.

With the center points located, the intersecting lines are center punched as shown in Fig. 94. As a test of the accuracy of the center location, the piece may be placed between centers and caused to rotate by the left hand and noting the high spots with a piece of chalk held in the right hand. This operation is shown in Fig. 95. The error, if any, is corrected by inclining the center punch and moving the center over, as in Fig. 96. This is continued until the necessary accuracy is obtained. The center punch marks are then enlarged by giving the punch a good hard blow with the hammer, thereby insuring their being drilled in this location.

These marks are then drilled to specified size with a small drill followed by a 60 degree countersink, Fig. 97, or by using the combination drill and countersink, Fig. 98. This operation is performed on a drill press or in the lathe as shown in Fig. 99.

When production is required for center-drilling work, a regular centering machine is used. On small runs the lathe can be so arranged to accomplish the same results. Fig. 100 is a typical set up. When this method is used it is not necessary to layout the center as its location is governed and guided by the use of a self centering chuck and the center drill being in alignment with it. Some examples of good and bad center drilling practice are shown in Figs. 101, 102, and 103.

Mounting Work Between Centers

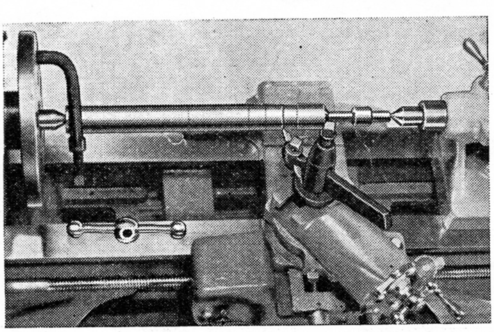


Fig. 104. Work properly mounted between the lathe centers and ready to machine.

LESSON 23

WITH the centers properly located and drilled in the stock, the operator must select a lathe dog (see Lesson 2) of proper size and shape for the job at hand. Attach this dog to one end of the work with the tail pointing away from the work in such a way as to act as a driver when positioned in the slot of the work driving plate. Remember that finished surfaces must be protected with a piece of copper or soft brass under the set screw so that the finished surface will not become damaged.

The tailstock is next brought up to an ap-

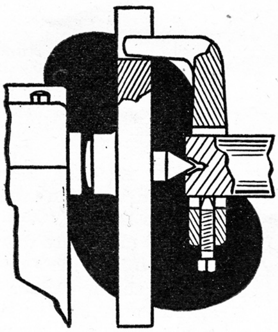


Fig. 105. Incorrect mounting, with dog too short, riding in bottom of slot.

proximate position allowing enough space with some room for adjustment to permit the length of stock to be placed between the centers and easily removed.

Next put some center lubrication such as white or red lead in the center hole of the tailstock end of piece of work being turned and place the headstock center hole on live center and tail of dog in driving slot as shown in Fig. 104, then by turning the tailstock hand wheel cause the dead center to enter the remaining center hole. Only enough pressure should be brought to bear at this point to permit the stock to rotate freely. As the stock turns friction may cause the hole to wear larger and sometimes heat from machining operations will cause expansion necessitating occasional adjustment of this pressure and the addition of more center lubrication.

Before turning on the machine, it is best to make a visual examination of the driving dog in the slot to note whether it is clear and not riding in the bottom as shown in Fig. 105. If this should be the case it is necessary to secure a larger dog and the situation will usually be remedied.

The work is now properly mounted and ready for machining.

Tool Bits Need Skillful Grinding

LESSON 24

BEFORE doing any actual machining operations on the lathe, the operator must become acquainted with the various shaped tool bits as described in Lesson 6 and learn how to grind them. See also Figs. 106 an 107.

We will consider here the grinding of the tool bits for straight turning as all other bits are similarly ground with only slight modifications for the job they are to perform.

Let us assume that we have an unground tool bit blank as shown in Fig. 108 of high speed steel and wish to grind a cutter bit for turning such as we see in Fig. 109.

The tool bit blank is always held in the hands of the operator and should never be held in the tool holder while grinding as the bit may get too hot, causing it to lose its temper or result in forming minute cracks on the cutting edge which would then break down when the bit is put into service. Also, the tool holder may take severe punishment by the careless operator who grinds part of it away.

The operator should secure a pair of goggles or other protection for his eyes and then dress the grinding wheel. Finally, he should set the tool rest so it is not more than 1/8″ away from the wheel.

With a good stance in front of the grinder, the operator brings the tool blank into position with the wheel so as to cause the material to be removed at the proper

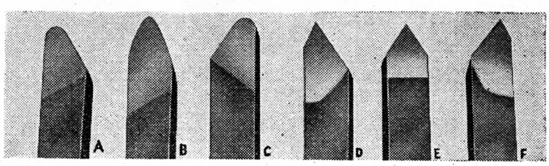


Fig. 106. Set of ground cutter bits. A, lefthand turning. B, round nose. C, righthand turning. D, lefthand side. E, threading. F, righthand side.

FIG. 9 DATA FOR THE MACHINING OF VARIOUS MATERIALS

MATERIAL	Front Clearance	Side Clearance	True Back Rake	Side Rake	Cutting Speed Feet Per Minute	LUBRICANT
STEEL (MILD COLD ROLLED)	8°	8°	16½° TOOL HOLDER ANGLE	22°	80 to 100	DRY OR CUTTING OIL
STEEL (MEDIUM HARD)	8°	8°	12°	14°	60 to 80	CUTTING OIL
STEEL (TOUGH ALLOY)	8°	8°	8°	12°	50	CUTTING OIL
CAST IRON	8°	8°	5°	12°	50	DRY
STAINLESS STEEL	8°	10°	16½°	10°	40	CUTTING OIL OR SULPHURIZED MINERAL OIL
COPPER	12°	14°	16½°	20°	120	DRY OR CUTTING OIL
BRASS	8°	8°	0°	0°	200 to 300	DRY OR PARAFFIN OIL
BRONZE	8°	8°	0°	2°	80 to 120	DRY OR PARAFFIN OIL
HARD BRONZES	8°	12°	10°	2°	30 to 80	DRY OR PARAFFIN OIL
ALUMINUM	10°	8°	35°	15°	200 to 500	DRY OR KEROSENE
MONEL & NICKEL	10°	13°	8°	14°	100 to 120	SULPHURIZED MINERAL OIL
MOLDED PLASTICS	12°	8°	0°	0°	100 to 120	TURN DRY THREAD WITH CUTTING OIL
CAST PLASTICS	14°	10°	0° to -5°	0°	150 to 200	TURN DRY THREAD WITH CUTTING OIL
FORMICA & MICARTA	15°	10°	16½°	10°	200	DRY
FIBER	15°	12°	0°	0°	80	DRY
HARD RUBBER	20°	15°	0° to -5°	0°	200	DRY OR COOLANT

These Angles Are Recommended by Competent Authorities. All Angles Are TRUE Angles, Measured from Horizontal and Vertical Planes. (See Figs. 1, 2 and 3 Above.) Speeds Are Correct for Standard High Speed Tool Bits. Special Alloy Bits Permit a Somewhat Higher Cutting Speed.

Fig. 107. "Lathe Cutting Tools" is a large blueprint wall chart containing valuable reference data for machinist, apprentice and student.

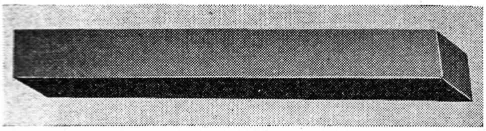


Fig. 108. An unground cutter bit.

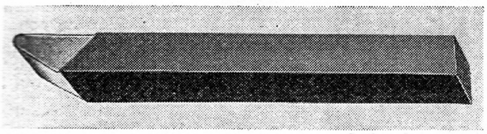


Fig. 109. A cutter bit after grinding.

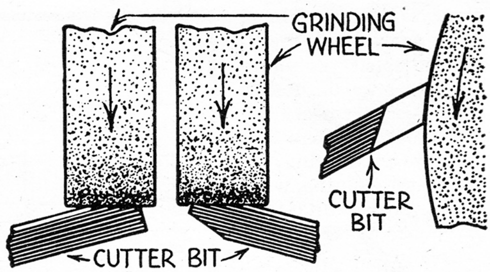


Fig. 110. Fig. 111. Fig. 112.
Fig. 110. Grinding left side of cutter bit.
Fig. 111. Grinding right side of cutter bit.
Fig. 112. Grinding front of cutter bit.

place. The tool should be moved back ana forth across the whole face of the wheel thereby causing the wheel to wear evenly. Care should be exercised at this point so as not to run off the wheel and grind the operators fingers. Do not exert too much pressure as otherwise grinding cracks may develop.

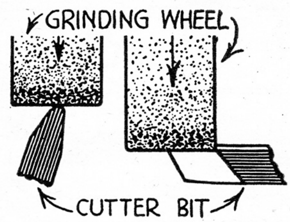


Fig. 113. Rounding the end of a cutter bit.

Fig. 114. Grinding side rake and back rake.

If at all possible, all grinding of high speed steel bits should be performed on a wet wheel. If this is not possible, care should be taken not to overheat the tool and then quench it in water as this sudden and extreme change in temperature also will cause the bit to crack along the cutting edge and break down in service.

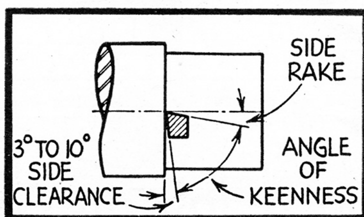


Fig. 115. Correct side clearance and side rake of cutter bit.

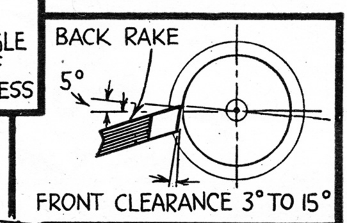


Fig. 116. Correct front clearance and back rake of bit.

See Figs. 110, 111, 112, 113, and 114 for position of tool while grinding the leading edge, side and front clearance and rake angles. Also Figs. 115 and 116 show these angles with reference to set up while turning. These angles can be checked with a regular gage or protractor while the tool bit is in the holder. In general, the clearance and rake angles are greater for machining the mild steels than for the hard steels. Some materials such as brass require no rake and lead or babbit even a negative rake is desirable. See tool shapes for recommended clearances and rake angles for kind of material being machined. If the clearance angles are too great, the cutting edge will break down rapidly and if the clearance is not enough the resulting finish will be very poor or the tool may not cut at all.

Fig. 117. Honing the cutting edge of a bit with an oil stone.

Finally, after the tool has been ground to the correct shape with due consideration given to correct and adequate clearance and rake angles for the material being machined, the cutting edges should be honed as shown in Fig. 117. This honing prolongs the tool life and also results in a better finish with less heat being generated at the cutting edge.

How to Set Up the Tool

LESSON 25

AFTER the proper tool shape for the operation at hand has been selected, with the correct clearance and rake angles, and after the cutting edge has been honed, the operator may proceed to set it in the machine.

Fig. 118 shows how to set the tool bit in the holder and the holder in the tool post. Note the minimum of overhand at points A and B. Fig. 119 shows the top view of a righthand turning tool properly set for straight turning while Fig. 120 shows the detailed shape of a righthand turning tool. Figs. 121 and 122 show the height of the tool point with reference to

the various machining operations. While this is the accepted procedure in setting up the tool it may be necessary to make slight alterations due to peculiarities in the design of the work piece. Additional instructions with reference to setting the tool for boring will be described in the lesson on that subject.

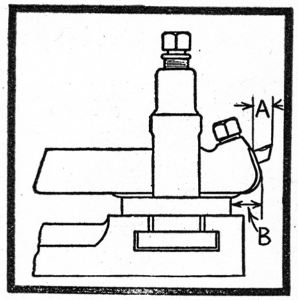


Fig. 118. Position of tool in tool post.

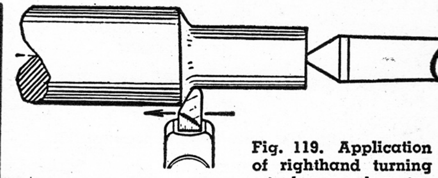


Fig. 119. Application of righthand turning tool, properly set.

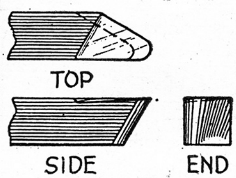


Fig. 120. Detailed shape of righthand turning tool.

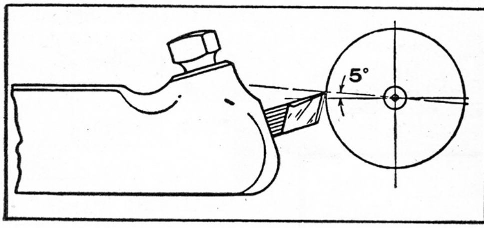


Fig. 121. Cutting edge of cutting bit is 5 degrees above center for straight turning.

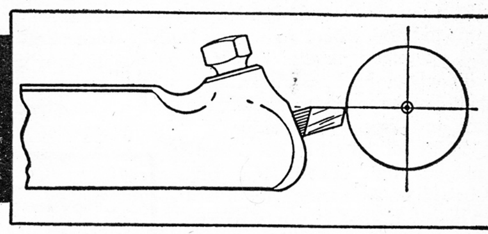


Fig. 122. Cutting edge is on center for thread cutting, taper turning, machining brass, copper, etc.

Facing a Job on Centers

LESSON 26

IN FACING a job on centers, the work is first mounted, then a facing tool, Fig. 123, is selected and set center high, See Fig. 124. The top view showing how to set this tool is clearly pictured in Fig. 125. With the tool properly set the operator measures the length of stock and notes how much excess material must be removed. Half of this excess material should be taken from each end.

A good method of machining off this material is by the step method as follows: First turn the tool bit in the clear and advance it in not to exceed 1/8 and by hand motion cause the tool to travel longitudinally toward the headstock the desired amount then cause the tool to travel toward the tailstock and advance the tool in another 1/8, then take this cut and other similar cuts until the piece is roughed down the desired amount. This method is known as "step" turning or facing. In order to get a good finish it is desirable to take a finish cut by power. The carriage should be locked in position when taking this cut so as to finish the end of the work square with its axis.

The other end is finished in a like manner after reversing the dog to the other end and marking off the length wanted. The rule laid along the work and the point of the tool will aid in getting the correct length.

The speed for facing is calculated from the outside diameter of the piece of work using the formula given in Lesson 5 with consideration given to kind of material and kind of tool. The power feed should be arranged so as to give the desired finish and all cuts made in an outward direction when finishing the ends of material on centers. For facing work held in a chuck or on a face plate due consideration will be given at the time chuck and face plate work is discussed.

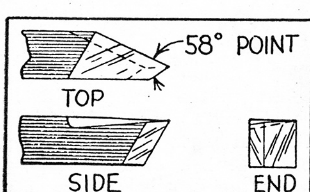


Fig. 123. Detail of righthand side tool.

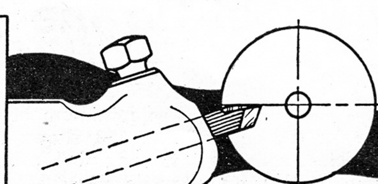


Fig. 124. Position of cutter bit for facing the end of a shaft.

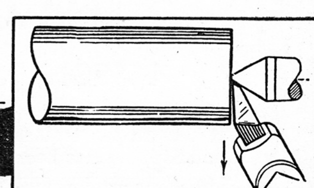


Fig. 125. Application of righthand side tool.

Straight Rough and Finish Turning

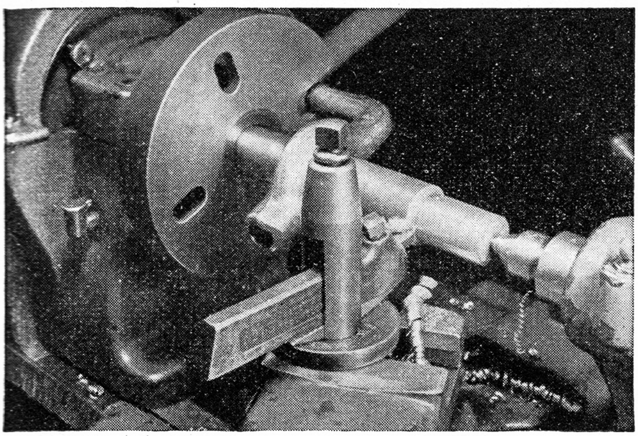


Fig. 126. Turning a steel shaft mounted between centers.

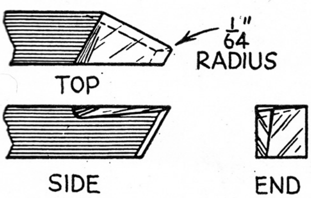


Fig. 127. Detail of roughing tool.

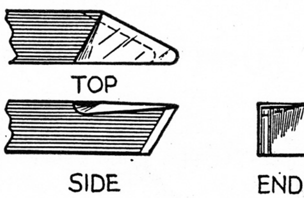


Fig. 129. Detail of finishing tool.

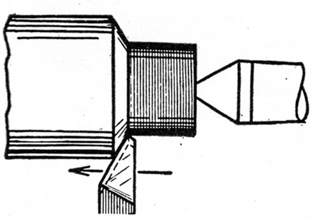


Fig. 128. Application of roughing tool.

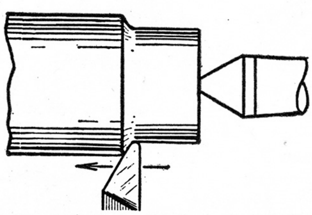


Fig. 130. Application of finishing tool.

LESSON 27

WHEN the work has been faced to length the center holes must be redrilled to the size desired. This is usually performed on the drill press using the combination drill and counter-sink center drill. With the centers redrilled and the work again mounted between centers, we are ready to perform our external cylindrical turning operations. Fig. 126.

First, we will assume that the operator selects the right-hand roughing tool, Fig. 127, and has it correctly mounted in the tool holder and the machine, Fig. 128. The next step is to calculate the speed using the formula "$R.P.M. = C.S. \times 4 \div D$" and set the belts or controls to obtain this speed as near as is possible on the machine that he is operating.

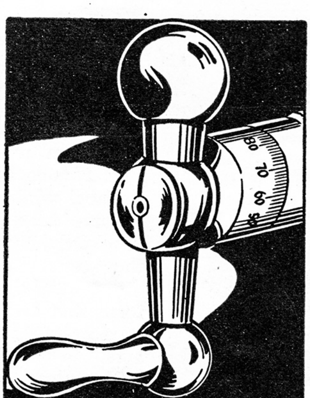


Fig. 131. Micrometer collar on cross feed screw of lathe.

The feed is next set and should be as heavy as is possible, being consistent with the diameter and length of work, also considering the size of work in relation to machine. If too heavy a cut is taken, it may cause the work to spring or bend or damage the lathe centers. If too light a cut is taken, then the machine is not being worked to capacity.

The roughing cuts are taken until the stock is within 1/32" or approximately .030" of finished dimension. Rough dimensions may be checked with the outside calipers as explained in Lesson 8. If the work is to be turned cylindrical the entire length, the operator should turn only one half, then place the dog on the other end and using the same tool setting turn the other end until the two cuts meet in the center. In this way the entire shaft is rough turned and all pent up stresses are released before the finish cuts are taken.

For the finishing cuts a finishing tool, Fig. 129, is set up, Fig. 130, and the speed is increased approximately 50 percent while the feed is cut down enough to produce the desired finish. After the cut is started having traveled an 1/8", tool travel is stopped and the work rotation stopped. It is then checked with calipers or micrometers, depending on the accuracy required. Any adjustments necessary are made at this time and the cut is completed in one pass to the center if the work is to be the same size from one end to the other. The work is then turned around and the other end finished, using the same tool setting.

The operator should be most cautious about removing the chips during any turning opera-

tions as they are very sharp and sometimes quite hot. So as to avoid injury in this connection chips are best removed by a chip rake or a pair of pliers. When machining work where the chips are small and tend to fly in all directions it is best for the operator to wear goggles or other safety protection.

The graduated collar, Fig. 131 can and should be used as an aid in setting the tool in the required amount when turning.

This collar is graduated in thousandths of an inch and is used similar to a micrometer. Here is where an understanding of the use of decimal equivalents described in Lesson 9 comes in handy. Let us take, for example, stock 1¼" in diameter in the rough and turn it to one inch. As we have ¼" of excess material to remove, or .250", we turn the tool in until it touches the work, then with the tool in the clear at the end of the work we advance it in .125 which is only one half of .250 as each one thousandth movement of the crossfeed causes twice that amount to be removed from the stock as it cuts all the way around. This method is used regardless of the amount necessary to be removed even though it is small.

Likewise, if when the start of a cut indicates that too much material is being removed the present reading is noted and the dial reversed a half turn to remove any play in the mechanism and then brought up to a number less than the preceding number by half the amount undersize. After another ⅛" travel of the tool the measurement is again checked. If correct, the cut is permitted to continue, otherwise additional adjustments are made until the correct reading is obtained.

Also the operator should become familiar with the dial on the compound rest, as it may vary in the number of divisions. It is used many times for securing lateral dimensions in decimals on thickness, lengths, etc.

Both collars can be set at zero at the start of cut or the operator may add to the present reading to obtain the ultimate setting.

How to Turn to a Shoulder

LESSON 28

SHOULDER turning is very much like straight turning. The setup of the tool is the same, and what has been said previously about speeds and feeds also holds true.

The first step in shoulder turning is to determine where the shoulder is to be and lay it off. The use of the hermaphrodite calipers, Fig. 132, is one way of locating this distance, and straight turning takes place up to a safe distance from the line. Another method of laying off shoulder distances is by using a necking or cut-off tool and making a slight recess as shown in Fig. 133, and taking cuts to this recess, Fig. 134.

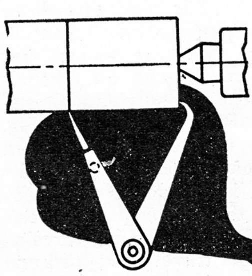


Fig. 132. Measuring with a hermaphrodite caliper to locate a shoulder.

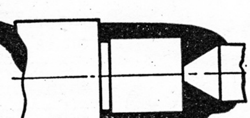


Fig. 133. Detail of a shoulder recess.

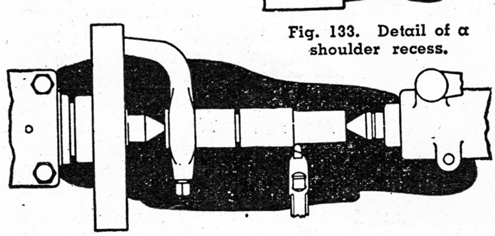


Fig. 134. Location of a shoulder marked with a parting tool before turning the diameter.

Shoulders are sometimes specified as sharp, round, or angular depending on the intended use of the part. If a sharp shoulder is wanted, this is shaped by a shouldering tool or recessed to obtain clearance for grinding. The round corner is usually shaped with a corner radius tool as shown in Fig. 135. While the round corner is the strongest, the piece part does not always permit it to be used. In using the radius corner tool, the operator will sometimes find that if he operates at the same speed as regular turning there will be undue chatter because so much of the cutting tool is in contact with the work. To remedy this, reduce the speed and apply cutting oil to the tool while machining. Filleted corners, that is what internal round corners are termed, are checked with radius gages, and the accuracy of the corner is dependent to a great extent on the tool shape. It is best to check the tool form first with the radius gage and to exercise care in setting the tool. Both are essential for accurate work.

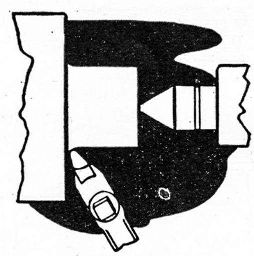


Fig. 135. Finishing a shoulder with a fillet.

Shoulder turning may be summed up as the process of turning the surface between two diameters to a definite shape, such as, square, recessed, or filleted to fit the job.

Filing, Polishing and Lapping

LESSON 29

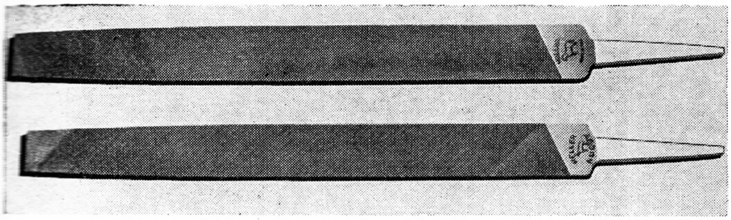


Fig. 136-A. A mill file (upper one) with single cut teeth and long angle gives a shear cut. A mill file (lower) designed with a special pattern double cut is used for lathe work, too, and with good results.

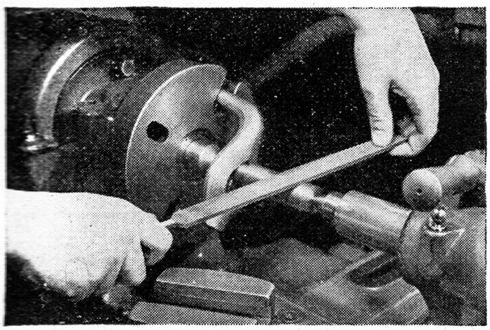


Fig. 136-B. Correct position for filing to remove tool marks.

ORDINARILY, most finish-turned surfaces are smooth enough without additional treatment, if the cutting tools have been properly ground and set along with proper speed and feed. Sometimes, however, there may be a slight taper on the work that needs correcting. But if the work is very close to size, another cut is out of the question as this may result in undersize. The correction is made by filing. The file is also frequently used for rounding or beveling corners and removing sharp corners.

Filing in the lathe must be performed properly or it will do more damage than good. A few suggestions to follow in order to achieve good results are:

Select a mill file (Fig. 136-A, upper file) with single cut teeth preferably having a long angle so as to give a shear cut. The single cut file will not "pin" as quickly as an ordinary double cut file. The lower file in Fig 136-A, with a special pattern double cut of teeth, is used extensively in lathe and polishing work where a smooth finish is desired. It is made in three cuts, bastard, second, and smooth.

In filing, increase the speed of the lathe about 50 per cent more than for regular turning of the same diameter work. It is highly recommended that the operator learn to file left-handed. It is just as easy to acquire the habit at the start and many manufacturers require left-handed filing of lathe work as a safety measure.

Clean the file frequently with a file card or brush.

Apply chalk to file occasionally as this will prevent pinning to a great extent.

Hold the file at a slight angle, Fig. 136-B, using a long steady stroke, lightening up a bit on the return, and moving over only half the width of the file for each stroke.

Filing should be done cautiously, in lathe work, as it is practically impossible to produce absolutely true cylindrical surfaces. It is better to spend a little more time in setting your cutting tools to do the job accurately. Do not depend on filing to do that for you.

Polishing on the lathe, Fig. 137, is frequently resorted to in order to produce a highly polished finish. The speed is also increased for this work as it was in filing. The abrasive used for polishing is known as emery cloth (aluminum oxide) and can be had in varying degrees of fineness and coarseness. The resulting finish depends upon the selection of grit size. If the starting surface is fairly rough, coarse cloth is first used and followed by the finer grades until the desired finish is obtained. Emery cloth comes in sheets or in economy rolls of various widths. The latter is best adaptable to lathe polishing. A small quantity of oil added to the surface of the cloth will improve its cutting and the lasting quality of the finish. And remember that finish is a test of one's workmanship.

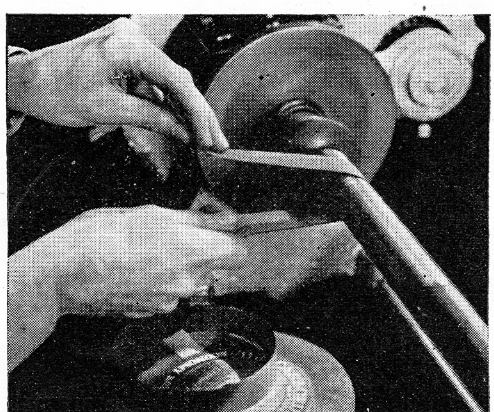


Fig. 137. Polishing with emery cloth.

The cloth should not be held in one place too long but moved back and forth along the work so as to prevent it from cutting rings. Because of the high speed used in polishing, the tail center must be set so the work revolves freely and lubrication should be applied frequently.

A speed or polishing lathe should be used for all small diameter pieces of work.

Lapping in the lathe, Fig. 138, is a polishing method as well as a sizing operation. The lap may consist of a strip of emery cloth attached to a shaft, or it may be made of lead, copper, cast iron, or other metal. It can be performed either internally or externally on hard or soft material. An internal lap is shown in Fig. 139. The lap or the work may be caused to rotate, and it is charged or loaded with the desired grade of emery dust, diamond dust, or other abrasives to produce the required cutting action or finish. It is done at high speed.

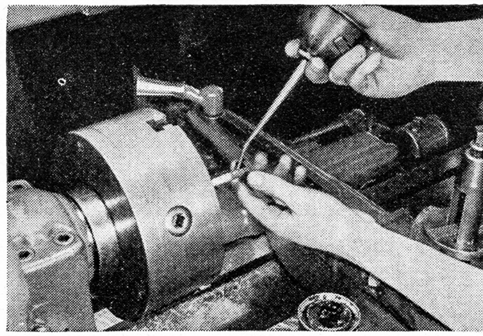


Fig. 138. Lapping is a polishing as well as a sizing operation.

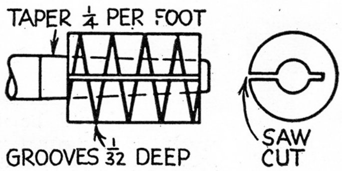


Fig. 139. A cast iron lap for internal work.

Lapping will result in a true cylindrical surface that is also correct in size as well as having a smooth finish. It is used internally in the making of drill bushing, bearings, and ring gages; and externally, in making plug gages and other true cylindrical shapes.

How to Do Knurling

LESSON 30

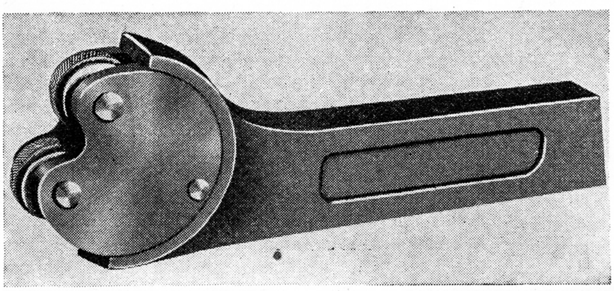


Fig. 140. Knurling tool for lathe.

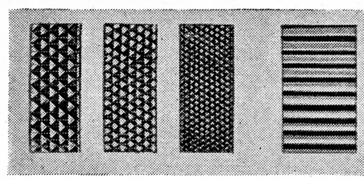


Fig. 141. Samples of knurling patterns.

KNURLING is the process of roughing a surface in order to provide a positive grip on parts requiring adjustment. The knurl shape or pattern may be either that of a diamond or straight lines and is caused by impressing a hardened tool of that shape into the work which causes the surface to become raised into that pattern. Knurling is also sometimes used to increase the diameter of a part to effect a press fit, or it may even be used for surface appearance.

The knurling tool is shown in Fig. 140, and specimens of choice in sizes of three diamond shaped knurls and one straight-line pattern are pictured in Fig. 141. The size and nature of the work is the determining factor in selecting the pattern or pitch to use.

The setup of the knurling tool, Fig. 142, is of great importance. The tool should be set so both rolls are forced into the work an equal amount and that the face of the

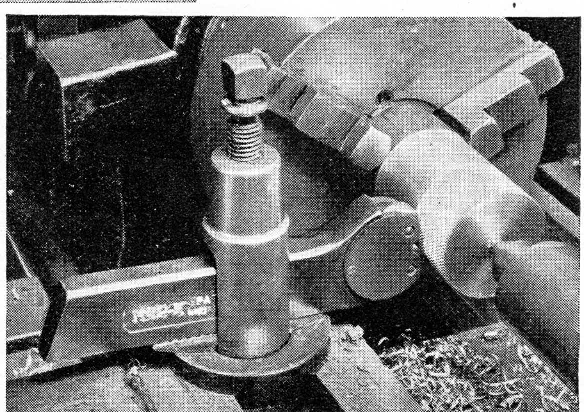


Fig. 142. Knurling a piece of steel in a lathe.

knurl is parallel to the work surface. As considerable pressure must be applied it is necessary to have the work well supported. The lathe speed should be set to about half that used for ordinary turning. With the setup properly made, the lathe is turned on and the knurling tool brought to bear lightly on the surface near the tailstock end. If the impression is incorrect in shape, it is best to move to a new spot and start over until the desired shape is secured. Cutting oil is applied as an aid in producing good knurled surfaces.

At the headstock end of the lathe, the cut is stopped by disengaging the clutch and with the work revolving the knurls are forced in a bit farther. Then reverse the direction of the leadscrew or the motor, thereby causing the tool to travel toward the tailstock. These operations are continued until the knurl takes on the proper appearance. Sharp points are desirable on some pieces while on others such as plug gage handles they are not desired.

Grooving and Cut Off Work

LESSON 31

THE cut-off tool shown in Fig. 143 may also be used for grooving. Its end shape may be varied to suit the job. This type of cutter is very convenient as it needs only to be ground on the end for clearance as the side clearance is already part of the cutter blank. It is held in a special holder. This type tool may also be made on the end of a regular cutter blank, but that necessitates considerable grinding.

The tool is set up as indicated in Fig. 144. Care is taken that the point is center high and that the tool is at 90 degrees to the work.

Grooving (or necking as it is sometimes termed) is the cutting of a recess at the termination of a shoulder as an aid when cutting threads or in grinding as it provides clearance for the tool or grinding wheel at the end of cut. Cutting off with this tool is accomplished in much the same way except the work must be held in a chuck or collet.

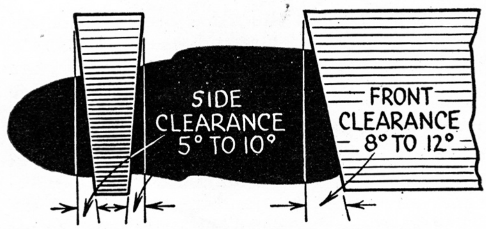


Fig. 143. Clearance angles of cut-off tool.

When using the cut-off tool and performing the above operations, the lathe should be operated at reduced spindle speeds to avoid chatter. Half turning speed is the usual procedure. While power feed may be used, hand feed is preferred by most operators. A liberal supply of cutting oil will aid in smoother operation on steel. The carriage should be locked in position once the tool is set and the tool fed into the work gradually but continuously until proper depth is obtained. The cross dial graduations may be used to determine the depth of cut when necking or the outside calipers, whichever the operator prefers. On deep grooves or large cut-off work, it is sometimes best to cut part way in and back the tool out moving it over and widening the cut slightly to prevent binding.

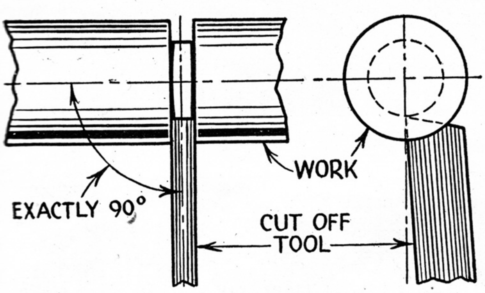


Fig. 144. Setting the cut-off tool into the work. The blade must be at a right angle to the work and the point should be on the exact center line.

Some operators grind a small chip curl on the cutting point of the tools. This should not be attempted by the beginner as he will waste much of his tool in each sharpening.

Grooving or necking may be termed the process of cutting a channel or furrow on a cylinder. The cutting of a channel or groove in a bored hole is usually referred to as the process of channeling or recessing. The shape of such grooves is determined by the shape of the tool and the depth it is fed into the work.

The three most commonly used shapes are the square, the round, and the "V" groove. The square and the round grooves are used most frequently at the termination of a threaded portion of a cylinder or against a shoulder that is to be ground. The "V"-shaped groove is used extensively on pulleys for the "V"-type belt.

The tool shape is classified as an end cutting type or form tool and clearance and use should be governed by these facts. Always remember to have the tool set center high, use slow speeds and use a liberal application of cutting oil.

The cutting off operation is performed with a cut off tool and is used many times for performing a shaping operation on the end of the work piece. Beveling and rounding operations can be incorporated by shaping the tool correctly and applying proper cutting clearance angles.

Watch Your Step on Chuck Work

LESSON 32

SOME work does not lend itself to support between centers while being machined. In such cases, the chuck is used for holding purposes, Fig. 145.

The chuck may be of any variety as described in Lesson 2. It is fastened on the spindle in place of the driving or face plate. Before placing a chuck on the spindle, be sure to clean the chuck threads as shown in Fig. 146. Apply a small amount of lubricating oil on the spindle and screw the chuck all the way up to the shoulder.

When the desired chuck has been selected for the job and properly placed on the lathe, the operator makes certain that the right jaws are used as each chuck has two sets, one for inside and the other for outside chucking. Some chucks use the same jaws by reversing them.

The universal chuck, Figs. 147 and 148, in which all jaws move in unison needs little explanation as to its use and application, but the independent chuck where each jaw must be set independently needs detailed explanations.

We will take, as an example, the chucking of a casting in a four-jaw independent chuck as shown in Fig. 149. Note the jaws are arranged to hold the casting away from the chuck face.

First, the operator measures the approximate size of the piece of work and sets each jaw by observing the concentric rings on the chuck body to a size that will admit the piece of work. The work is then placed against the jaws and each one tightened separately so as to grip the work. Now with a piece of chalk held in the right hand and the work rotated the chalk marks will indicate the high spot as shown at "A" Fig. 149. The jaw opposite this mark is loosened and the jaw at the chalk mark is tightened. Again a test is made repeating the above procedure until the piece runs true enough for the job at hand.

On cylinders that must be set up with absolute accuracy, a test indicator is used and the work set to rotate perfectly. Sometimes the high spot falls between two jaws. In that case the

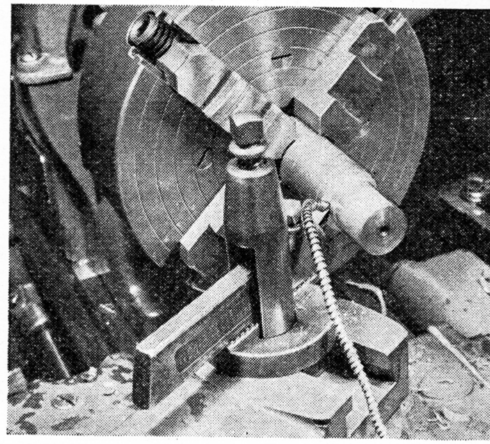


Fig. 145. Machining in an independent chuck.

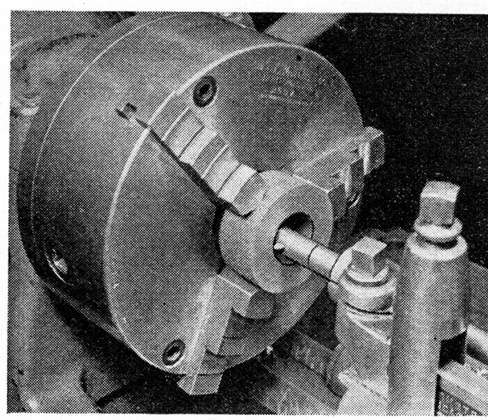


Fig. 147. Round work held in a universal chuck.

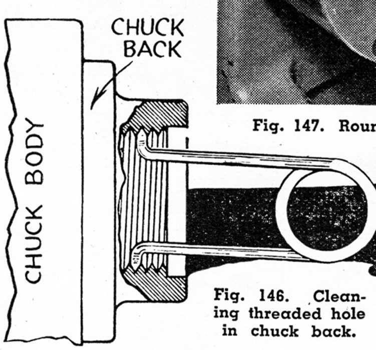


Fig. 146. Cleaning threaded hole in chuck back.

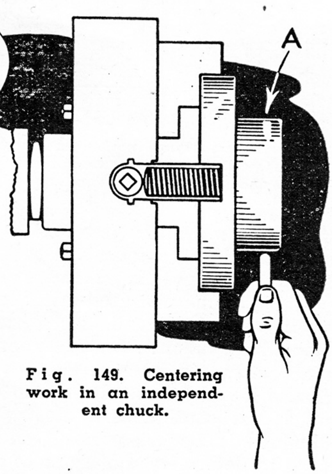


Fig. 149. Centering work in an independent chuck.

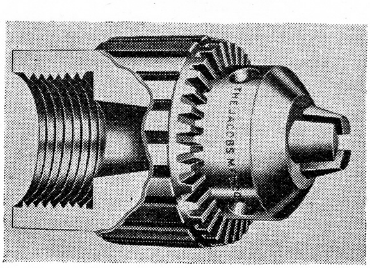


Fig. 148. Hollow headstock spindle chuck.

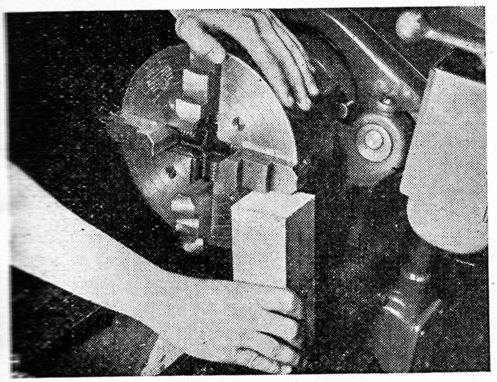


Fig. 150. Removing a chuck from a lathe spindle.

two jaws must be moved in and opposite jaws out a like amount.

Work should not be removed from the independent chuck until it is finished as considerable time is required to reset a piece after it is once started. In chuck work the piece should be gripped securely with as little overhang as possible. If another piece identical in shape and size is to follow only two jaws need be loosened and the new piece set up in a short time by tightening these two jaws.

When the operator is through using a chuck, he should carefully remove same by placing a block under one jaw at the back of the machine as shown in Fig. 150 and cause the machine to be rotated by hand (never use power). This will loosen the chuck slightly and it can easily be removed by unscrewing.

Sharpen Your Drills Accurately

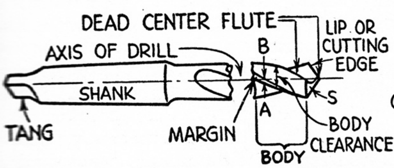


Fig. 151-A. Side view of a twist drill.

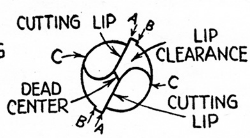


Fig. 151-B. End view of the point of a twist drill.

LESSON 33

EVERY lathe operator should know how to properly sharpen drills so they will cut correctly. The twist drill drawings, Figs. 151, 152, and 153, show the parts and cutting angles. As an aid in checking the point for its correct angle the rule and gage pictured in Fig. 154 is of considerable help. If the correct angles are not ground on this drill point the result will be as shown in Fig. 155 and in some cases the drill may even fail to cut, break, or dull very quickly.

Drills may be sharpened by machine to exact shape with proper clearances but very few small shops have this equipment so it is well that the machinist know how to grind drills by the hand method.

It is rather difficult to explain just how to go about this sharpening a drill by hand but the operator can get a fair idea through trial and error and by checking will soon learn the correct procedure.

Some tips on sharpening drills that should prove helpful are as follows:

Never rely on guess work, always check the drill point carefully with a gage.

Do not let the drill point get so hot that it will draw the temper from the drill.

Do not quench a hot high speed drill point in cold water as this will form minute cracks along the

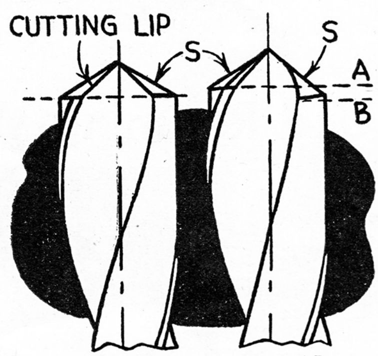


Fig. 152-A. Fig. 152-B.

Fig. 152-A. Drill without lip clearance. The cutting lip and heel, S, are in the same plane.

Fig. 152-B. Drill with proper lip clearance. Heel line, B, is lower than the cutting line, A. The distance between A and B measures the lip clearance.

Fig. 153-A. Drill point showing proper lip clearance angles at the circumference of the drill.

Fig. 153-B. End view of drill point showing proper angle between point and lip.

Fig. 153-C. Drill point with lips ground identically. Lips are of equal length, clearance and angle.

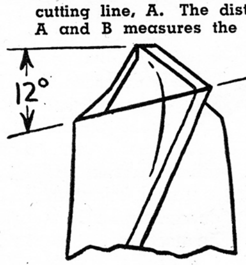


Fig. 153-A.

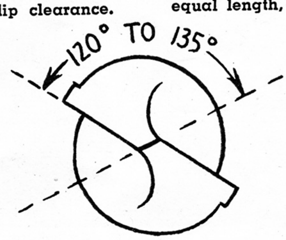


Fig. 153-B.

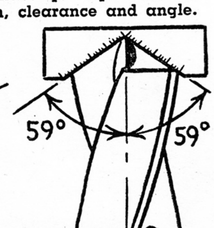


Fig. 153-C.

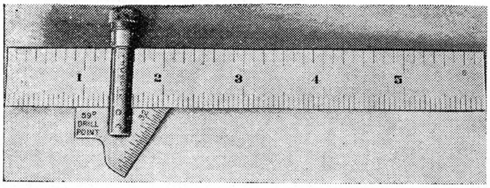


Fig. 154. Drill gage attachment for ordinary hook rule or straight steel rule. Checks both length and angle of drill lips.

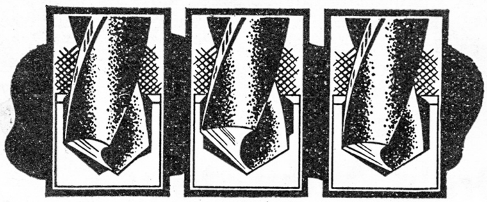


Fig. 155. Common mistakes in drill grinding. Note that in each case the resulting hole must be oversize. At left, lips of unequal angle and unequal length. Drill point actually travels around the center of the hole. At center, lips of unequal angle. The right lip does all the work. At right, lips of equal angle but unequal length causing excessive wear on the right lip.

cutting edge too fine to be seen by the naked eye but soon show up if the surface is etched with acid. The edge on such drills will break down in service before many holes are drilled.

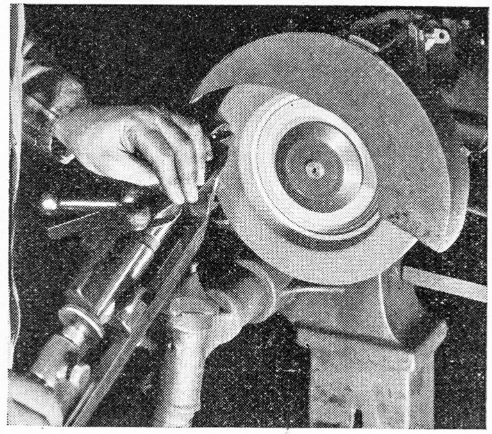


Fig. 156. Machine grinding is recommended for the accurate sharpening of drills.

Machine grinding, Fig. 156, is recommended as the more accurate method of sharpening drills. When properly machine ground, a drill will generally cut faster, last longer and produce more accurate holes than if ground by hand. Drills larger than ⅜" should be machine ground.

Some materials have different machining characteristics and require different clearance angles as well as increased or decreased included angle of the point. In general, hard materials require less clearance and greater included angle while just the reverse applies to soft materials.

Drilling and Reaming Work

LESSON 34

DRILLING in the lathe may be accomplished in two ways, (1) either by revolving the drill as shown in Fig. 157 and 158, and supporting the work on or in drill crotch or pad pictured in Figs. 159 and 160, or (2) by causing the work to rotate and holding the drill stationary in a drill chuck or in the taper spindle of the tail stock, Figs. 161 and 162.

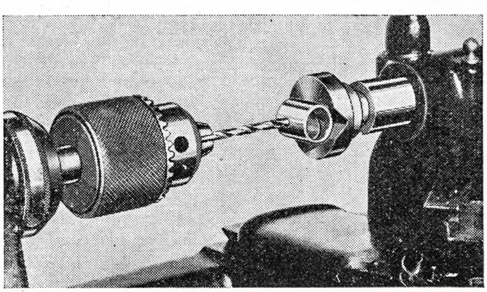


Fig. 158. Drilling an oil hole in a bushing with crotch center in tailstock.

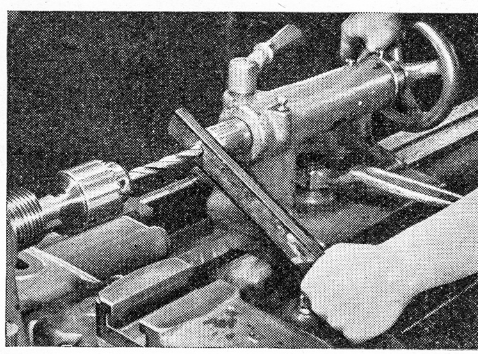


Fig. 157. Using the lathe as a drill press.

In both cases the drill is caused to feed into the work by turning the tailstock handwheel. The speed to revolve the drill or the work is governed by the diameter of the drill, kind of material the drill is made of and kind of material being drilled. The same formula discussed under Lesson 5, on speeds and feeds, may be used or reference made to any handbook on the subject.

Reaming operations in the lathe, Figs. 163 and

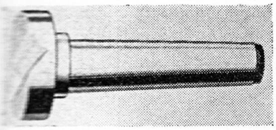
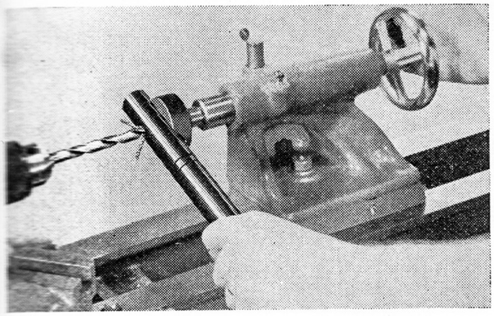


Fig. 159-A. Crotch center for use in tailstock of lathe.

Fig. 159-B. Drilling hole in a shaft with crotch center held in tailstock.



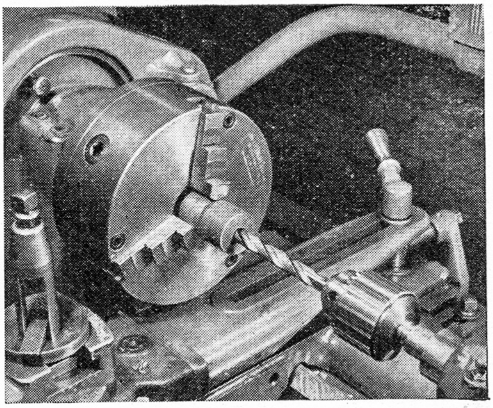


Fig. 161. Drilling work held in chuck.

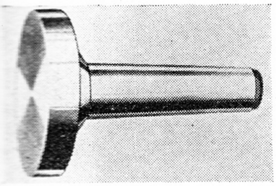
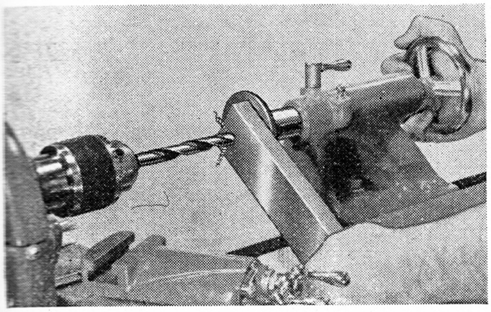


Fig. 160-A. Drill pad for use in tailstock of lathe.

Fig. 160-B. Drill pad in use. Hand feed is used to force work against drill.



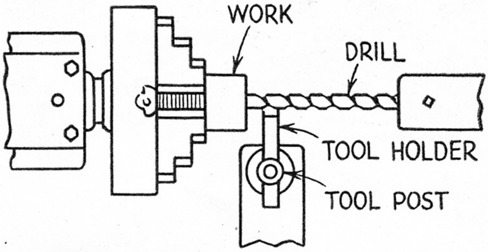


Fig. 162. Using tool holder in tool post to steady the drill point.

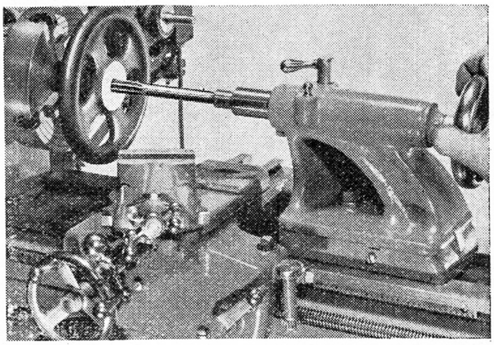


Fig. 163. Reaming in the lathe.

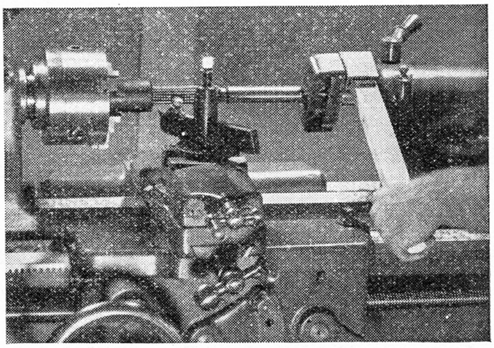


Fig. 164. Floating reamer driver.

164, are similar to drilling except that the work or reamer must be revolved at a slower speed. Only a small amount not to exceed .015 of an inch should be left for removal by the reamer. The reamer must be fed in at a fairly rapid rate to avoid riding too long in one place and causing it to cut oversize. All reamers should be checked for size with a micrometer before use and should be sharp.

When reaming and drilling in the lathe, the tailstock must be in alignment with the axis of the work, otherwise the result will be an oversized hole.

While it is much safer to hold a drill or reamer in a chuck or in the tailstock taper spindle, some sizes are too large for this means of support. One way in which the larger sizes may be held is by supporting them on the tailstock center and holding them against the center firmly, with the toolholder against a dog or drill holder so as to prevent their pulling into the work.

How to Do Boring

LESSON 35

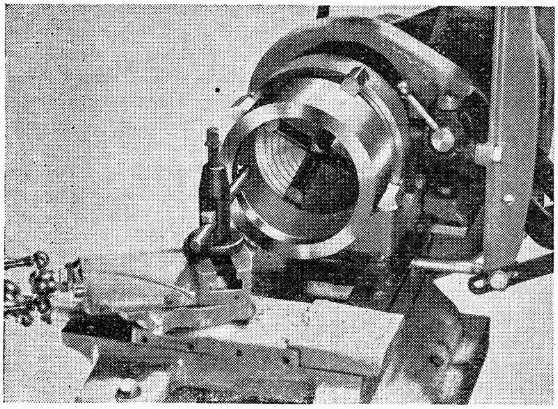


Fig. 165. Boring the inside of a large steel bushing. Note highspeed boring tool mounted directly in tool post for maximum rigidity.

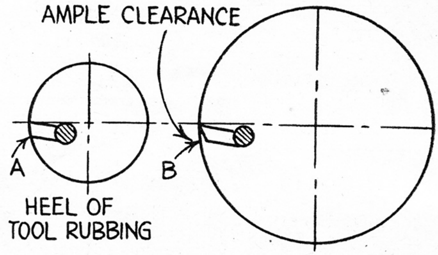


Fig. 166. This drawing shows how a certain angle of front clearance may be too small for one hole but satisfactory for a large hole. At A, the heel of the tool is rubbing. At B, in the larger hole, there is ample clearance.

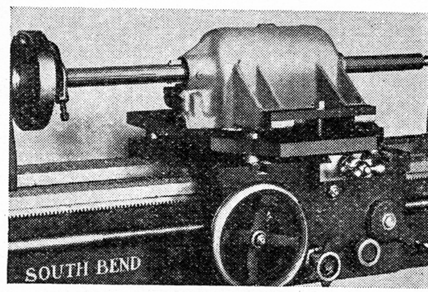


Fig. 167. Boring on the lathe carriage.

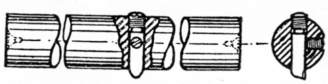


Fig. 168. Boring bar with a fly cutter.

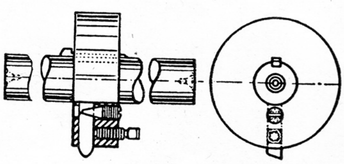


Fig. 169. Boring bar with boring head.

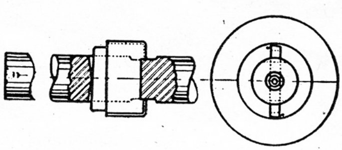


Fig. 170. Boring bar for sizing the hole.

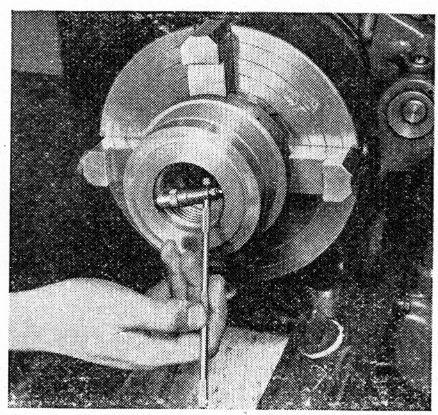


Fig. 171. Using an internal micrometer caliper for measuring the diameter of a hole.

BORING, Fig. 165, is the enlarging of a hole that must be a true cylindrical surface that has previously been drilled or cored. Boring is the only way to get odd size holes unless a special tool is made for the job. The tool used for boring may be forged or of the inserted cutter type. It is ground similarly to external turning except that the front clearance varies according to the diameter of the hole. It is greater as the size of the hole gets smaller. See Fig. 166.

Another method of boring large intricate castings is to clamp them to the carriage, Fig. 167, rotate the boring bar between centers and cause the carriage to move. Depth of cuts are regulated by setting the tool out as needed. These boring bars and adjustment features may be noted by examining Figs. 168 and 169, while Fig. 170 shows a sizing cutter that cuts on both edges.

Fig. 171 shows one method of checking a bored hole using an internal micrometer caliper. Another method is to use inside calipers as described in Lesson 8 and still another method is to use telescoping gages and checking these with outside micrometer calipers.

The secret of success in boring is to have the set-up rigid as possible, avoiding too much overhang in the tool and having a large enough bar so it will not spring away from the work.

Counterboring is similar to regular boring but only enters for a predetermined depth enlarging the mouth of a hole with its sides parallel to the axis of the hole and with its bottom at right angles to this same axis.

Countersinking is the enlarging of the mouth of a hole at an angle to the axis of the work.

Undercutting is the cutting of a recess at or near the back of a hole for clearance such as is necessary when doing internal threading.

Boring operations are really internal turning processes and are always resorted to when holes must be round or concentric. Another particular advantage is that holes can be finished to any given size and proper allowance can be left for reaming operations. Pick the right reamer, too, and keep it sharp.

When You Use a Mandrel

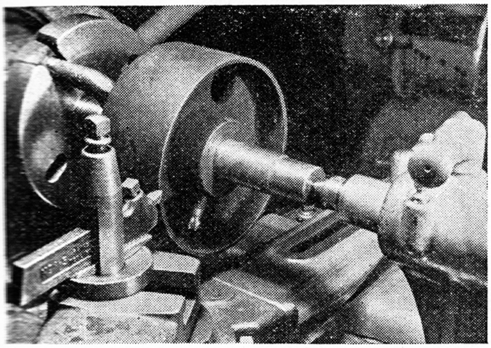


Fig. 172. Turning a pulley on a mandrel.

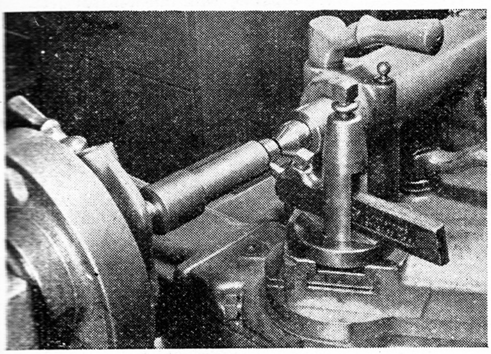


Fig. 173. Finishing a bushing on a mandrel.

MANDRELS are used as an aid in the turning of work that has a hole which has been bored or reamed and which needs further external machine operations, Fig. 172 and 173. The mandrel, Lesson 2, is forced into the hole on an arbor press, Fig. 174, and a lathe dog placed on the large end. The tailcenter of the mandrel is lubricated and with the work mounted on it, is placed between centers.

A drop or two of oil placed on the mandrel before placing it in the hole will aid in removing the work after machining and will prevent galling of the interior finish. All machining should be in a direction toward the large end of mandrel as this will tend to press the work on more tightly instead of causing slippage.

After machining, the mandrel is removed by pressing out large end first or down on an arbor press. Adequate support should be given close around the mandrel so as to avoid breaking of spokes, flanges, etc. as on pulleys and gears.

LESSON 36

The mandrel is sometimes called an arbor but this is considered incorrect terminology. The common mandrel is hardened and ground and is commercially manufactured in various standard sizes. The ends are centered and carefully lapped prior to grinding. The taper is usually .006″ per foot and the size is stamped on the large end. The nominal size is about one-third of the mandrel length from the small end. The ends of the mandrel are turned smaller than the body so that nicks and burrs on it, due to clamping of the dog on this surface will not injure the work or accuracy of the ground surface.

The mandrel should be considered a master tool and be treated as such. The operator should avoid cutting into the surface with the tool bit, protect its centers by keeping them clean and never use a steel hammer when inserting or removing the mandrel used in a piece of work.

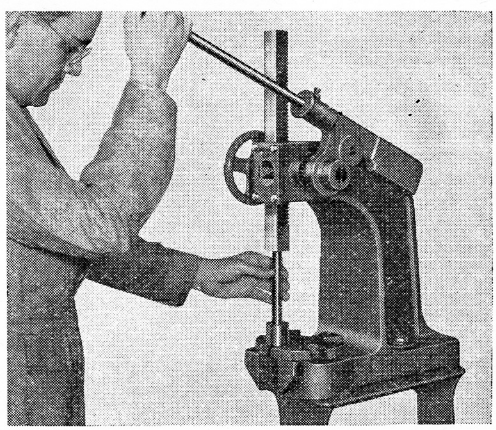


Fig. 174. Pressing a mandrel on a bushing before machining.

When using a mandrel it is especially important to have the live center run axially true, likewise the tail center must be in exact alignment with the live center for straight cylindrical shapes.

All turning and facing operations on mandrel work are similar to such operations performed on work held between centers, consequently tools are ground and set accordingly.

Besides the standard mandrels, the operator may make his own for in-between sizes.

How to Turn Tapers

LESSON 37

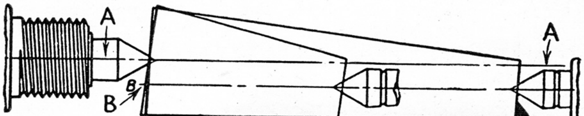


Fig. 176. With tailstock set over the same amount, pieces of different lengths are machined with different tapers.

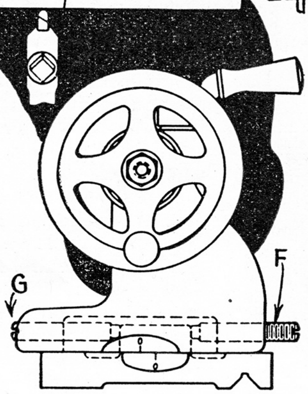


Fig. 177. Tailstock top set off center for taper turning.

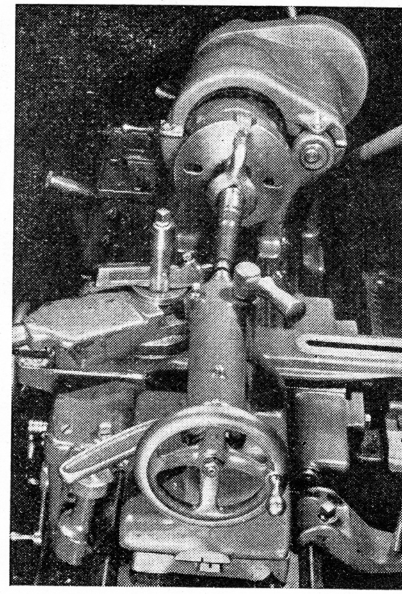
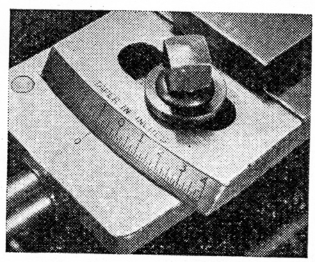


Fig. 175. Tapering with tailstock set over.



Fiig. 181. Graduations in inches per foot on the swivel bar.

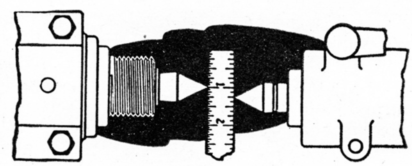


Fig. 178. Measuring the set over.

BOTH internal and external cylindrical tapers and angles can be turned on the lathe almost as easily as straight turning can be done. What has been said about cutting tools and their set up is equally true when it comes to turning and boring tapers.

The three most common methods of cutting tapers are: (1) tailstock set over method, (2) the taper attachment method, and (3) turning taper with the compound rest method. A fourth method sometimes used for short tapers is accomplished by using a square nosed tool set at the required angle.

The tailstock set over method, Fig. 175, requires no special attachments as it is a readily available method that can be used on any lathe. With the tailstock set either before, Fig 176, or behind, the center line of the lathe, the tool still travels parallel to the axis of the lathe and the result is a taper on the work piece. The amount of offset, length of the work and the direction of offset governs the size and small end of the taper

A simple formula for figuring the amount of set over is: Offset equals taper per inch $\times$

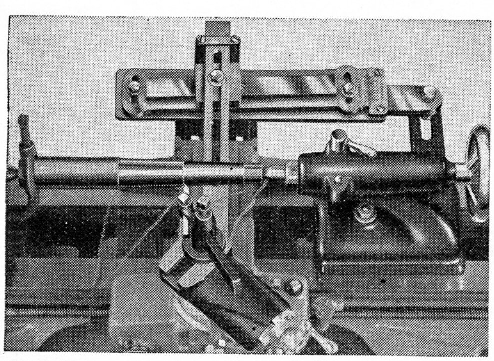


Fig. 179. Taper turning with taper attachment.

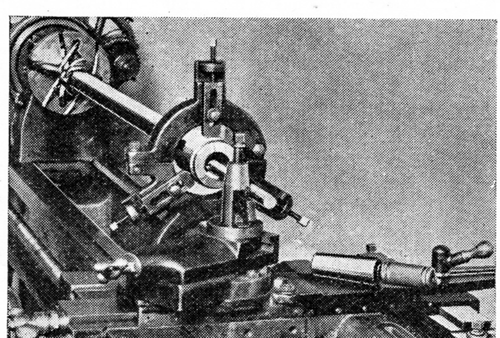


Fig. 180. Boring a taper with taper attachment.

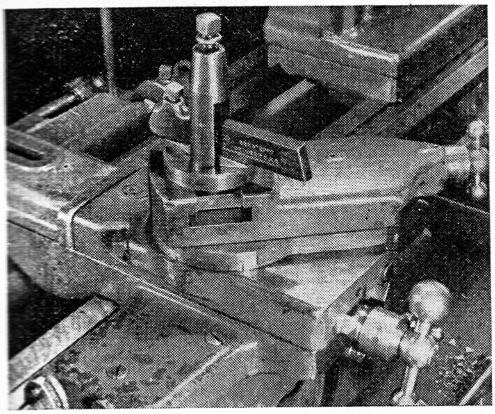


Fig. 182. Compound rest set at an angle for taper turning and boring.

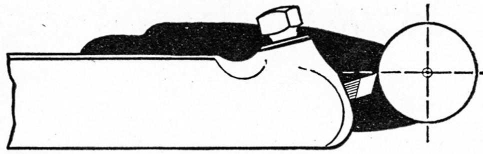


Fig. 183. Lathe tool cutter bit set on center for taper turning.

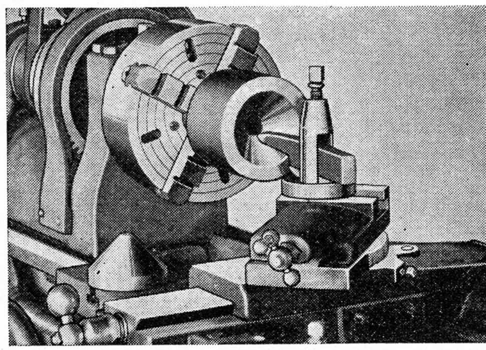


Fig. 184. Machining a conical punch and die with compound rest.

length in inches ÷ 2. Let's take an example: The specifications call for a taper per foot of .623 (Standard No. 4 Morse Taper). Divide this by 12 to find the taper per inch or approximately .052" per inch, multiplying this by the length 6" equals .312 and dividing this by 2 gives .156 set over. As .156 is the decimal equivalent of 5/32 this is the amount to set over the tailstock. See Figs. 177 and 178 for accomplishing this set over. After taking a cut, the taper should be checked and re-adjusted if necessary until it conforms to specification. (See Lesson 14, on tapers.)

This tailstock set over method has many limitations and has recently been dropped in favor of the taper attachment method. Here's one objection: When several pieces of approximate length need the same taper it is necessary to re-adjust for each piece for the slightest variation in length or the depth of center holes. Another objection is the unequal bearing and wear on the center holes. Also, after turning taper the centers must be realigned before the operator can turn straight again. This method is also limited to work turned between centers which eliminates external and internal tapers on chuck work.

The taper attachment method Figs. 179 and 180 eliminates all of the above limitations and is now the accepted method of producing both external and internal tapers. Fig. 181 shows an end view of the taper attachment. This is graduated in degrees on one end and in fractional parts of an inch on the other end and may be readily set or adjusted by the operator. All piece tapers using this method will be uniform regardless of the length or work. For setting taper attachment and checking tapers see Lesson 14.

The third method of cutting tapers, especially steep angles, is accomplished for both internal and external tapers by using the compound rest, Fig. 182. In using this method the angle of the taper must be figured or given in degrees and the compound rest set to this angle. The tool must be set on center as shown in Fig. 183 for this method as well as for all other taper turning methods. Fig. 184 shows a set up for boring and internal taper using the compound rest set at the required angle.

Steep angles are usually checked with a protractor for approximate angles and for more exact work the sine bar or other methods utilizing trigonometry is necessary.

Using the Steady and Follower Rests

LESSON 38

THE steady rest, Fig. 185, and the follower rest, Fig. 186, are used on the lathe in supporting long slender work or work that can be machined in no other way without support. They may be used in combinations as shown in Fig. 187 or separately as in Fig. 18.

Use of the steady rest is shown in Figs. 189 and 190 and the method of holding the driving end of the work is pictured in Figs. 191 and 192.

The procedure of mounting the work center or steady rest is as follows: Place the rest on the lathe in the position where it will permit the machining operations to take place if externally turned and near the outer end for all internal operations. Clamp it into position and open the top part to receive the work. Clamp the work

on one end in the chuck or attach a dog to it and tie same with raw hide lacing to the driving plate, Fig. 192. On work driven between centers this fastening operation is not necessary. Carefully adjust the jaws so the work will revolve freely. They are then tightened in place and the bearing surfaces lubricated with white or red lead or special lubricant.

The follower rest is applied in much the same manner as the steady rest except that it is fastened to the carriage and is located directly behind the point of the cutting tool and follows along as the cut is being taken, thereby giving sup-

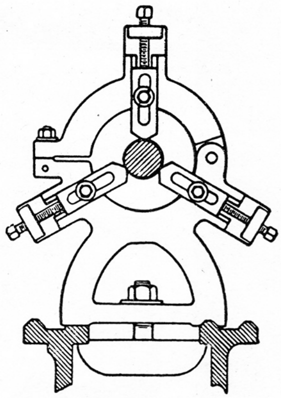


Fig. 185. The center rest mounted on the lathe bed.

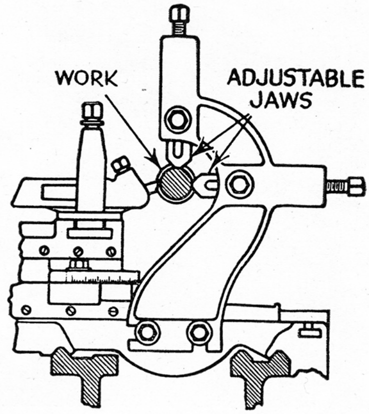


Fig. 186. The follower rest mounted on the lathe saddle.

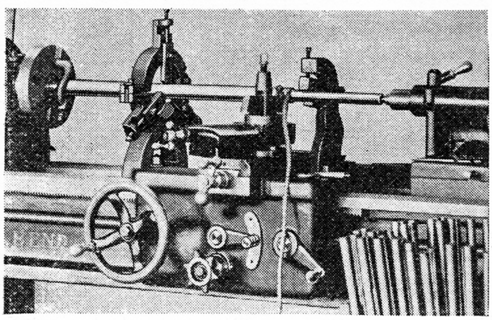


Fig. 187. Using both the center rest and the follower rest to support a long, slender shaft.

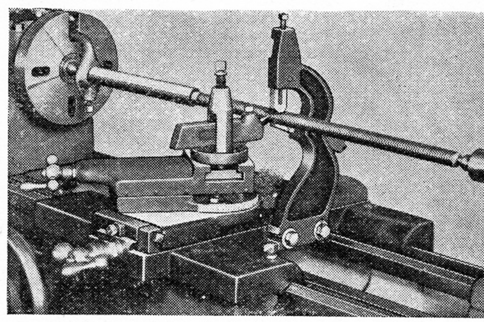


Fig. 188. Threading a long, slender shaft with the aid of a follower rest.

port where it is most needed.

The cut is started on the end and then the jaws are adjusted to ride on this finished surface. Adjustment should be made carefully and a lubricant applied to these bearing surfaces. The follower rest is especially useful when cutting threads on a long slender shaft. No doubt the operator can see other applications for both accessories either used singularly or together.

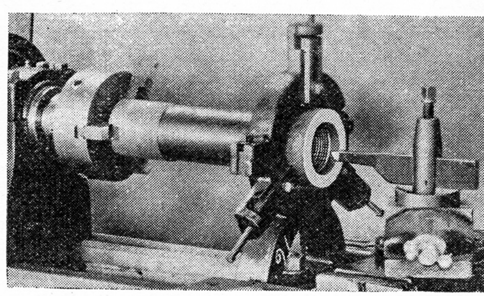


Fig. 189. Cutting an internal screw thread with a center rest.

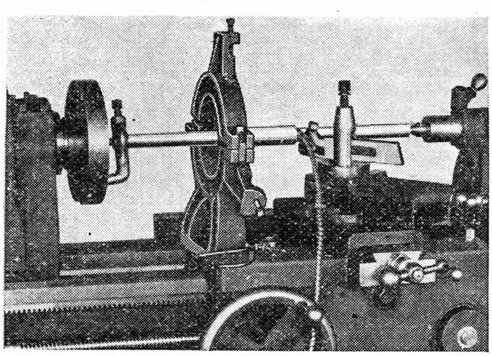


Fig. 190. Supporting a slender shaft with the center rest.

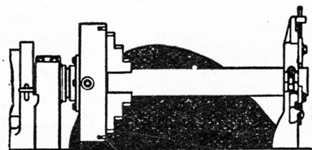


Fig. 191. Work mounted in chuck and center rest.

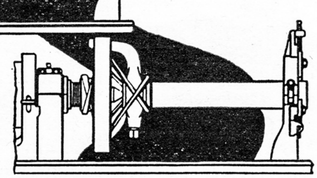


Fig. 192. Work mounted on center and in center rest.

Cutting Threads Will Test Your Skill

LESSON 39

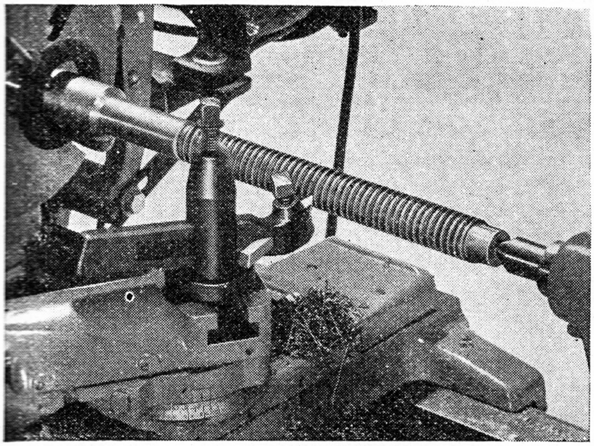


Fig. 193. Cutting screw threads in the lathe.

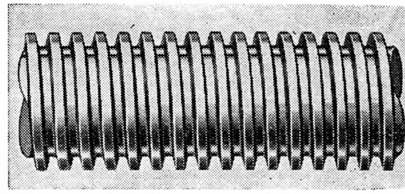


Fig. 194. Acme screw thread.

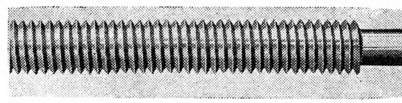


Fig. 195. American national coarse thread.

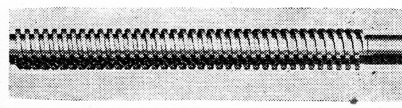


Fig. 196. Double square thread.

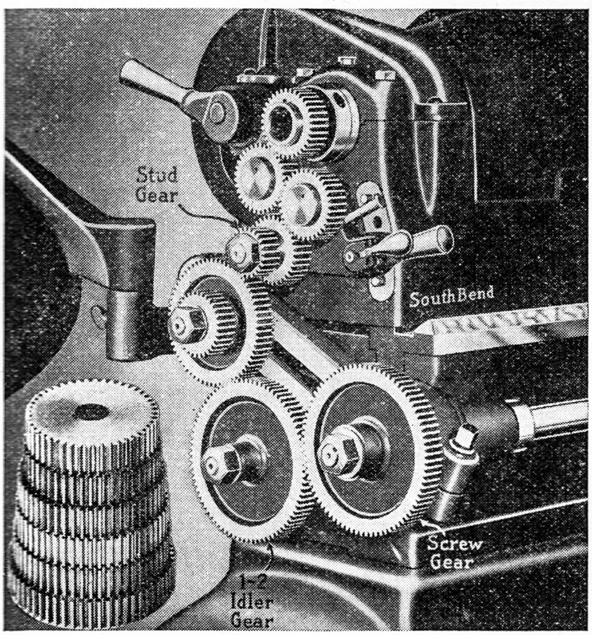


Fig. 197. Standard change gear setup for cutting screw threads.

Setting the Machine

THREAD cutting is a challenge to the skill of any lathe operator. Consider first the setting of the machine:

Fig. 193 shows a typical threading job being performed on the lathe while Figs. 194, 195, and 196 show various forms of threads. (See Lesson 15 on thread terminology.) The lathe may be set up to cut threads with standard change gears as shown in Fig. 197 and by noting the chart attached to the lathe and pictured in Fig. 198, the operator can quickly select the right combination of gears and their arrangement. Variation in feeds are obtained on some lathes by this method.

Another method of setting up other lathes to cut screw threads and adjust feed is accomplished through a gear change box. Fig. 199 and the accompanying chart Fig. 200 show the possible number of threads and feeds that can be obtained by proper setting of the various levers.

Both methods operate on the same principal, that of obtaining a relationship between the speed that the work is revolving and travel of the carriage and the tool. This ratio is changed by the various combinations of gears.

When cutting threads, the split nut is closed and the clutch is not used for the feed control. The split nut makes a more positive drive and provides for exact stopping at the end of a thread when the split nuts are disengaged as well as exact alignment when the thread dial is used according to instructions at the start of a thread.

The theory of how and why threads can be cut on a lathe is based on the number of threads per inch on the lead screw and the number of threads per inch desired on the piece being threaded. From this a ratio is worked out with change gears.

For example: if the lead screw had 6 threads per inch and 6 threads per inch was wanted on

the shaft, then even gears on the spindle stud and on the lead screw would do the job. If 12 threads per inch is wanted on the work with the same lead screw, then we would want the work to make two revolutions to one of the lead screw. Therefore, we would cut 12 threads if we use a ratio of two to one in our change gears placing the large one on the lead screw and the small one on the stud. Intermediate or idler gears can be of any number of teeth without affecting the ratio. Likewise any number of threads can be cut and the change gears calculated and set in position even though the chart may be missing from the lathe.

Setting the Threading Tool

Assuming that the gearing has been arranged, the operator's next job is to set up the tool. If sharp V or American national standard form threads are to be cut, the tool must have an included angle of 60 degrees and may be checked with a center gage as shown in Fig. 201. For American national standard screw threads the point must be straight across a different amount for each number of threads per inch. This tool must be ground to fit the gage Fig. 202. Not only must the tool be the correct shape but it must have adequate front and side clearance as shown in Fig. 203. A form thread cutting tool wherein the 60 degree is always correct and needs only an occasional grinding on top and can be adjusted is pictured in Fig. 204.

SCREW THREADS AND POWER FEEDS

10" SERIES "S" STANDARD CHANGE GEAR LATHE

THREADS PER INCH	STUD GEAR	IDLER GEARS	SCREW GEAR	CROSS FEEDS	LONG. FEEDS
4	24	FIG. 1	48		
4½	24	FIG. 1	54		
5	16	FIG. 1	40		
5½	16	FIG. 1	44		
6	16	FIG. 1	48		
6½	16	FIG. 1	52		
7	16	FIG. 1	56		
7½	16	FIG. 1	60		
8	32	FIG. 2	32		
9	32	FIG. 2	36		
10	32	FIG. 2	40		
11	32	FIG. 2	44		
11½	32	FIG. 2	46		
12	32	FIG. 2	48		
13	32	FIG. 2	52		
14	32	FIG. 2	56		
16	24	FIG. 2	48		
18	24	FIG. 2	54		
20	16	FIG. 2	40		
22	16	FIG. 2	44	.0056	.0152
24	16	FIG. 2	48	.0051	.0139
26	16	FIG. 2	52	.0048	.0129
27	16	FIG. 2	54	.0046	.0124
28	16	FIG. 2	56	.0044	.0119
30	16	FIG. 2	60	.0041	.0111
32	32	FIG. 3	32	.0039	.0105
36	32	FIG. 3	36	.0034	.0093
40	32	FIG. 3	40	.0031	.0084
44	32	FIG. 3	44	.0028	.0076
46	32	FIG. 3	46	.0027	.0073
48	32	FIG. 3	48	.0026	.0070
52	32	FIG. 3	52	.0024	.0064
54	32	FIG. 3	54	.0023	.0062
56	32	FIG. 3	56	.0022	.0060
60	32	FIG. 3	60	.0021	.0056
64	16	FIG. 3	32	.0019	.0052
72	16	FIG. 3	36	.0017	.0046
80	16	FIG. 3	40	.0015	.0042
88	16	FIG. 3	44	.0014	.0038
92	16	FIG. 3	46	.0013	.0036
96	16	FIG. 3	48	.0013	.0035
104	16	FIG. 3	52	.0012	.0032
112	16	FIG. 3	56	.0011	.0030
120	16	FIG. 3	60	.0010	.0028
160	16	FIG. 3	80	.0008	.0021

AUTOMATIC POWER FEEDS THROUGH FRICTION CLUTCH IN INCHES PER REVOLUTION OF HEADSTOCK SPINDLE

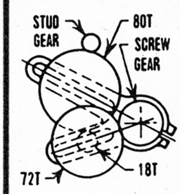


FIG. 1

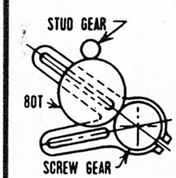


FIG. 2

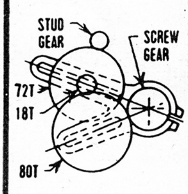


FIG. 3

Fig. 198. Change gear chart for standard change gear lathes.

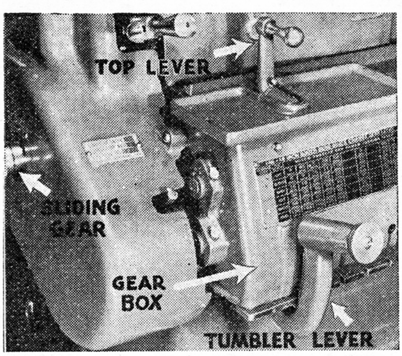


Fig. 199. Quick change gear mechanism for cutting screw threads.

16-INCH SOUTH BEND QUICK CHANGE GEAR LATHE

SLIDING GEAR	TOP LEVER	THREADS PER INCH—FEEDS IN THOUSANDTHS										AUTOMATIC CROSS FEED EQUALS .375 TIMES LONGITUDINAL FEED
IN	LEFT	4 .0841	4½ .0748	5 .0673	5½ .0612	5¾ .0585		6 .0561		6½ .0518	7 .0481	
	CENTER	8 .0421	9 .0374	10 .0337	11 .0306	11½ .0293		12 .0280		13 .0259	14 .0240	
	RIGHT	16 .0210	18 .0187	20 .0168	22 .0153	23 .0146		24 .0140		26 .0129	28 .0120	
OUT	LEFT	32 .0105	36 .0093	40 .0084	44 .0076	46 .0073		48 .0070		52 .0065	56 .0060	
	CENTER	64 .0053	72 .0047	80 .0042	88 .0038	92 .0037		96 .0035		104 .0032	112 .0030	
	RIGHT	128 .0026	144 .0023	160 .0021	176 .0019	184 .0018		192 .0017		208 .0016	224 .0015	

Fig. 200. Index plate for threads and feeds on quick-change gear lathes.

When cutting threads, the point of the tool must be set center high as shown in Fig. 205 and the tool should be flat on top having no back rake or side rake. Some experienced mechanics do use side rake for quicker cutting but it is best for beginners to refrain from this practice.

The tool is checked for proper alignment with the aid of the center gage for external threading as shown in Fig. 206 and for internal threading as shown in Fig. 207.

Of course the thread dial and thread stop are used if the machine is so equipped (as described in Lesson 3 on attachments) and the compound rest is set at 29 degrees or 30 degrees, Fig. 208.

The above instructions must be strictly followed if exact threads are to be the results.

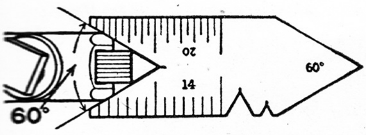


Fig. 201. Cutter bit for cutting screw threads is ground to 60° center gage.

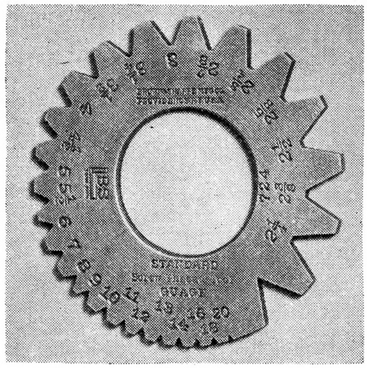


Fig. 202. Standard screw thread tool gage for grinding thread-cutting tools.

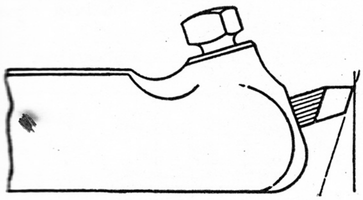


Fig. 203. Side view of lathe tool cutter bit ground for cutting screw threads.

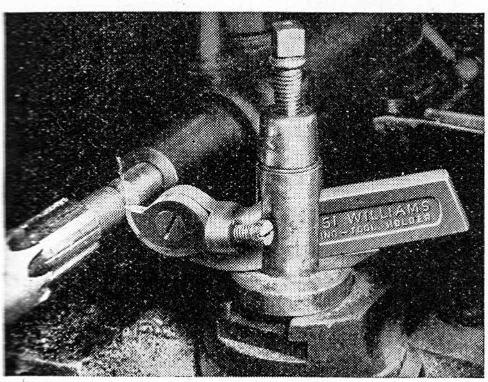


Fig. 204. Formed thread-cutting tool, solid type.

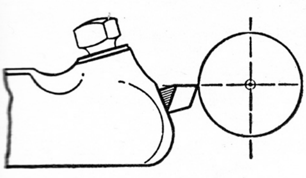


Fig. 205. Top of cutter bit set on center for cutting screw threads.

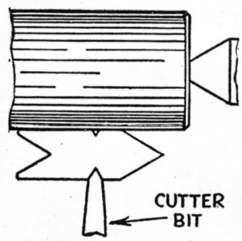


Fig. 206. Cutter bit set square with work for cutting external screw threads.

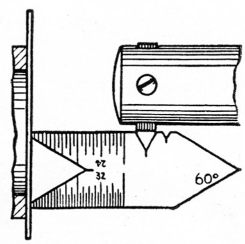


Fig. 207. Cutter bit set square with work for cutting internal screw threads.

Cutting Threads and Checking

With the machine properly geared and the tool set up, the operator is ready for the actual operation of cutting a thread.

The spindle speed for threading is usually about one-third that used for regular turning so the back gears will probably have to be used to get the proper speed setting. The first cut, Fig. 209, is taken and checked with a scale, Fig. 210, or with a screw pitch gage, Fig. 211, to ascertain whether the proper gearing has been used. At the end of the thread the split nut is disengaged and the tool withdrawn and brought back to the starting position turned in to the thread stop, Fig. 212, and the compound rest feed in approximately .005 and the split nuts engage at the proper time as noted on the thread dial. If no thread dial is available the split nuts are not disengaged, instead the tool is brought back to the start of the thread by reversing the motor or driving mechanism. This method is sometimes used when cutting threads only a short distance.

The feeding in of the tool at an angle, shown in Fig. 213, is continued until the thread takes shape, diminishing the amount of feed as the chip gets wider and as the thread begins to take form. It is good practice to take a free cut once in a while, that is with no feed, this will aid in producing a smoother finished thread. Also thread-cutting oil should be applied on all cuts after the first trial cut. Feed the tool straight in for the last few turns to give the thread a better appearance.

As the thread nears its correct size and shape, cuts are stopped and the piece checked with a thread pitch micrometer which reads the pitch diameter. This size can be readily found in any handbook for the various sized standard threads.

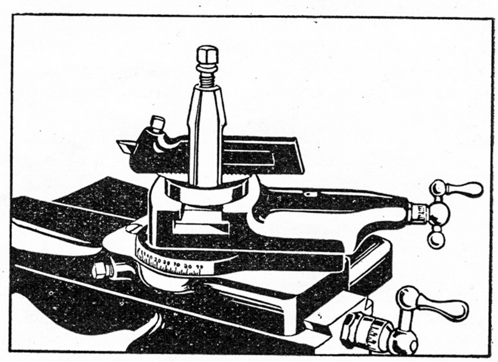


Fig. 208. Compound rest set at 29-degree angle for cutting screw threads.

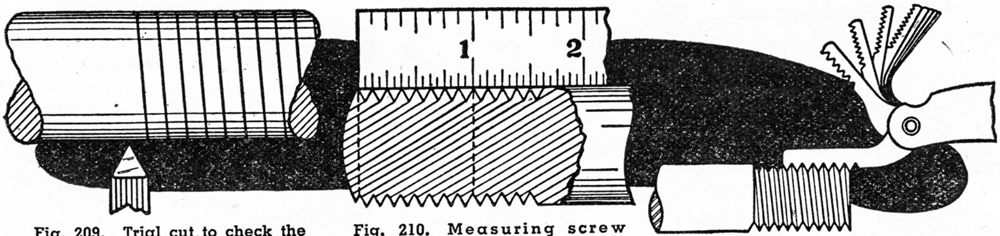


Fig. 209. Trial cut to check the setup for thread cutting.

Fig. 210. Measuring screw threads.

Fig. 211. Screw thread pitch gage.

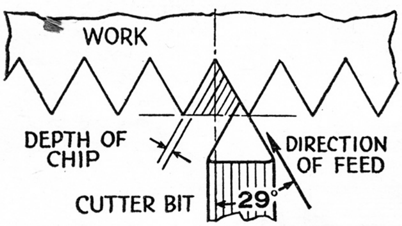


Fig. 213. Action of thread-cutting tool when compound rest is set at 29-degree angle.

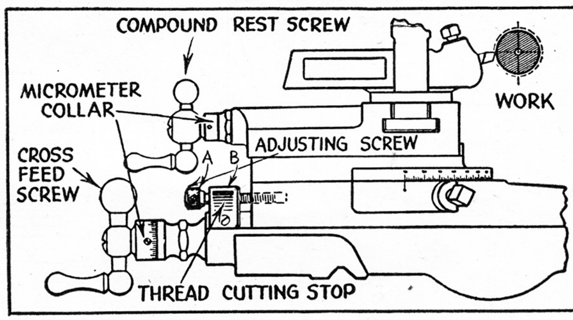


Fig. 212. Thread cutting stop attached to dovetail of saddle.

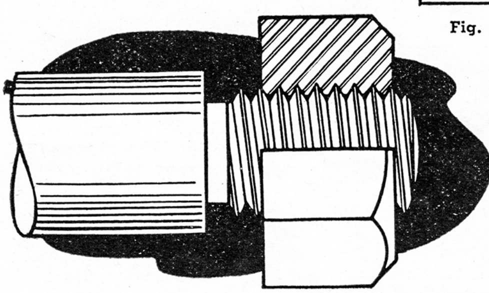


Fig. 214. Screw thread fitted to nut.

If no thread micrometer is available, the part may be checked with a thread ring gage or even a standard nut as indicated in Fig. 214. Other forms of threads are cut in like manner, the only difference being in the shape of the tool. With a little practice the operator will become quite proficient in thread cutting on the lathe.

The ability to produce the various shaped threads both rapidly and accurately on the lathe with a single point cutting tool is considered a valuable asset to the operator.

Collets Speed Up Small Work

LESSON 40

THE draw-in collet attachment is a quick and accurate device for chucking and holding small stock of symmetrical shape. Fig. 215 shows its use, while Fig. 216 shows a cross-section of the collet in place through the spindle of the headstock.

The draw-in collet chuck attachment consists of the spring collet, collet sleeve, spindle nose cap, and draw-in bar, with handwheel attached. Fig. 217 shows another type of step chuck that can be used with the draw-in bar for holding larger diameter work.

Collets can be obtained in sizes from 1/16″ to 1″ in diameter in steps of 1/64″. These are considered standard, while wire gage, number size, squares, hexagon and octagon shapes and sizes are considered special and are made up on order.

Some collets are made with a rough knurled finish on the inside as an aid in gripping the stock.

Collets should never be over or under expanded

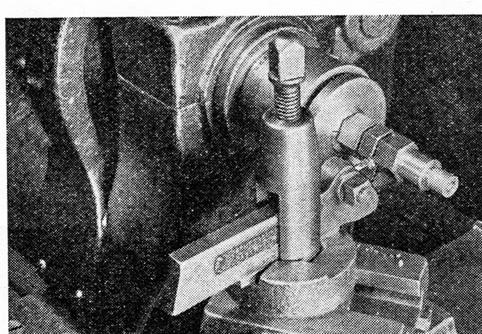


Fig. 215. Machining a part held in a draw-in collet chuck.

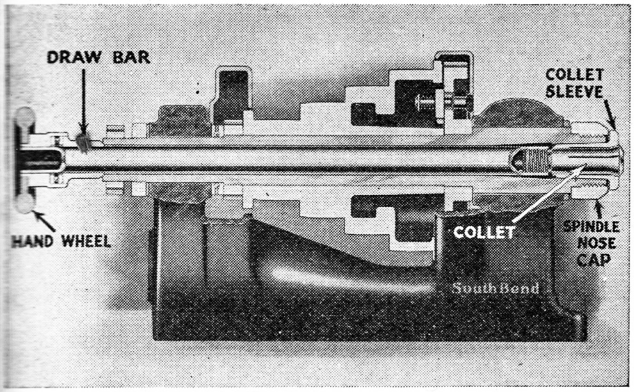


Fig. 216. Cross-section of headstock showing construction of draw-in collet chuck attachment.

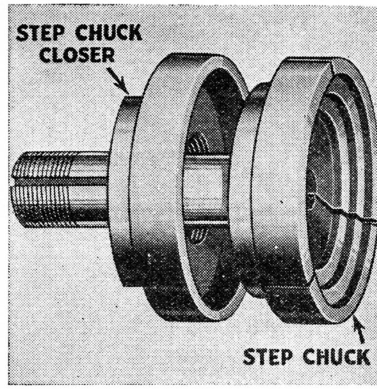


Fig. 217. Step chuck and closer.

more than one thousandths of an inch or their accuracy will be impaired. Care must also be exercised in keeping them and the work clean and free from burrs. Also rough stock should not be held in collets. It is better to use a universal chuck in this case.

How to Do Face Plate Work

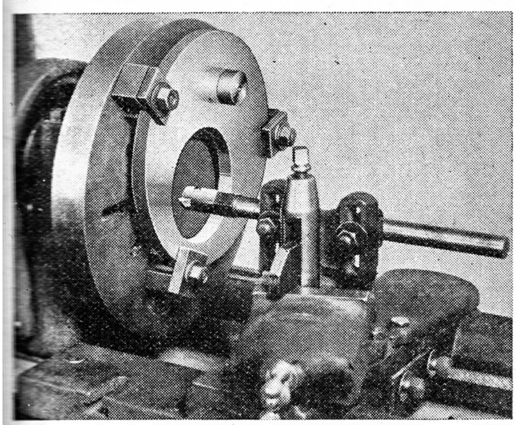


Fig. 218. Boring an eccentric hole on the face plate of a lathe.

LESSON 41

SOME work does not lend itself to being held and turned between centers, or being machined while being held in a chuck. The only way to hold it is to clamp it to the face plate, Fig. 218, or support it in a fixture which in turn is bolted to the face plate, Fig. 219.

When work is fastened to the face plate, it is necessary to locate it accurately. Fig. 220 shows a method of accomplishing this location. Often the work must be clamped in such a way that it is all to one side and this results in an out of balance set which necessitates a counter balancing arrangement such as pictured in Fig. 220.

When a fixture is located on the face plate, duplicate parts can be quickly located and machined. This type of setup is often used to good advantage on production work.

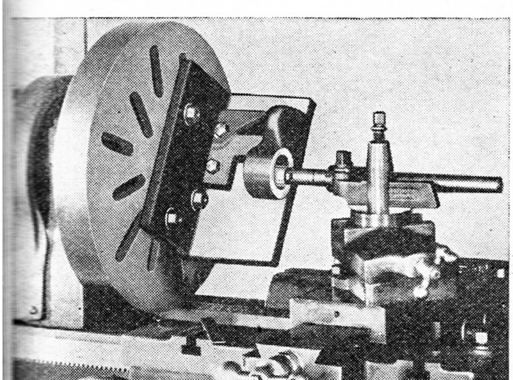


Fig. 219. Boring a bracket with an angle plate attached to the face plate.

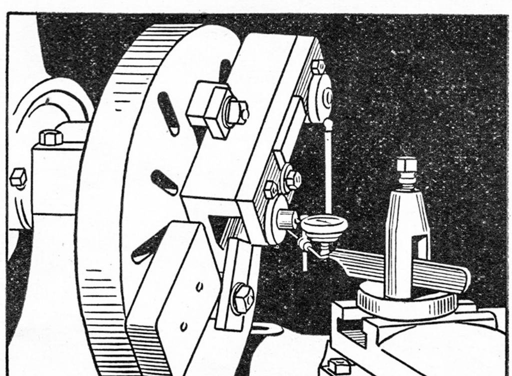


Fig. 220. Locating work on the face plate with a dial indicator.

Doing Milling on a Lathe

LESSON 42

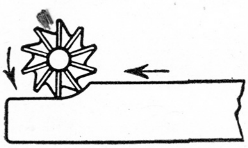


Fig. 221. Direction of feed for milling operations.

MILLING operations such as end milling, cutting keyways, and cuttting gear teeth can be done on the lathe. Note the set-ups pictured in Figs. 221, 222, 223, 224, and 225. And Fig. 226 shows an assortment of cutters readily adapted to this type of work. Each serves a special task.

By using a lathe for performing these small milling operations, the more expensive milling machine is released for larger work. Also, in some shops where a milling machine is not at hand, the work might have to be done outside causing delay and considerable expense.

The cutter is mounted in the spindle with a taper shank or held on an arbor between centers, and the attachment is fastened to the cross slide in place of the compound rest. The cut is controlled by the in-and-out cross feed, the longitudinal feed, of the lathe, and the vertical feed is accomplished in the attachment itself. Various angular adjustments are also possible.

The cutting of gear teeth is performed in much the same manner as on regular milling machines using an index arrangement.. Any good handbook will furnish the operator with the information necessary to perform this operation. Be sure to have this setup as rigid as possible.

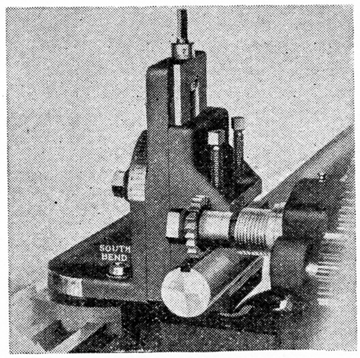


Fig. 222. Milling a standard keyway in a shaft.

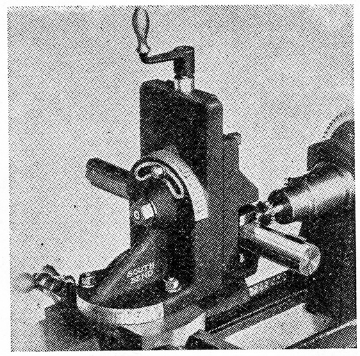


Fig. 223. Milling a Woodruff keyway in a shaft.

When using the gear cutting attachment, which in reality is a dividing head, many milling operations other than the cutting of gears can be performed, i.e., graduating, splining, slotting, cutting squares, hexagonals, octagonals, etc. Fixtures can be made that will facilitate many setups for production and in this way increase the versatility of the lathe as a milling machine.

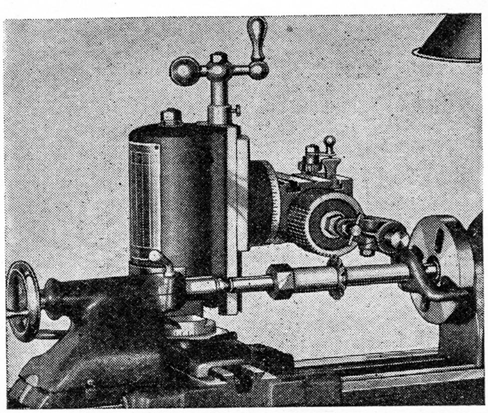


Fig. 224. Gear-cutting attachment.

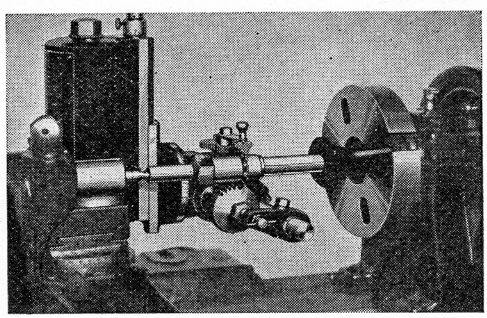


Fig. 225. Cutting a gear on a lathe.

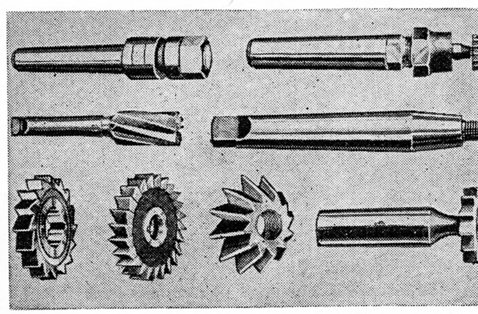


Fig. 226. An assortment of milling cutters and arbors used in lathe work.

External and Internal Grinding

LESSON 43

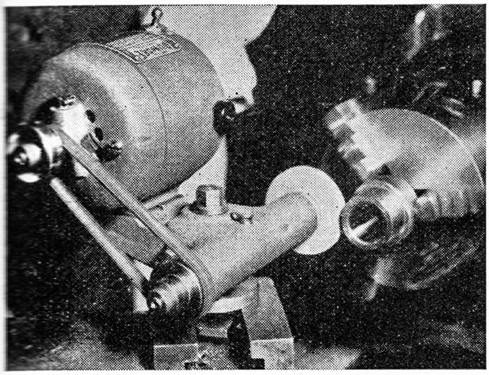


Fig. 227. External grinding attachment for a lathe.

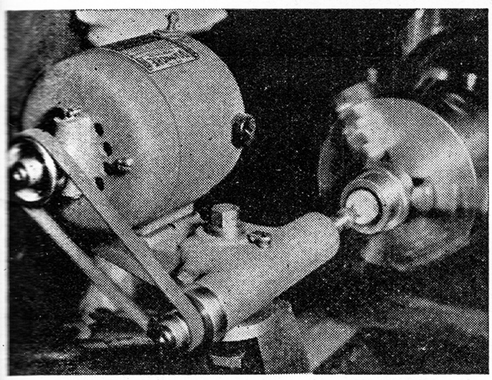


Fig. 228. Internal grinding attachment for a lathe.

THE lathe can also be used to perform many grinding operations.

The attachment in this case is a tool post grinder, shown in Fig. 227, set up for external grinding and in Fig. 228 set up for internal grinding. Also milling cutters, Fig. 229 and reamers, Fig. 230 as well as other tools can be ground in the lathe.

Grinding in the lathe should not be attempted without first covering the ways and bearings with some protection against the abrasive particles which would soon work into these parts and cause excessive wear. Also the operator must wear goggles to protect his eyes.

In grinding, the main consideration is that the proper wheel be selected for the kind of work to be ground and that it be operated at the recommended speed for its size, Fig. 231. It is also important to have the wheel dressed with a diamond dresser and properly guarded, Fig. 232.

Grinding is a sizing operation wherein only a few thousandths of material is

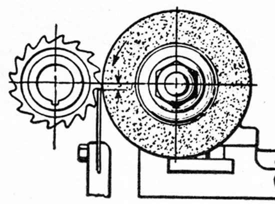


Fig. 229. Grinding clearance on a milling cutter.

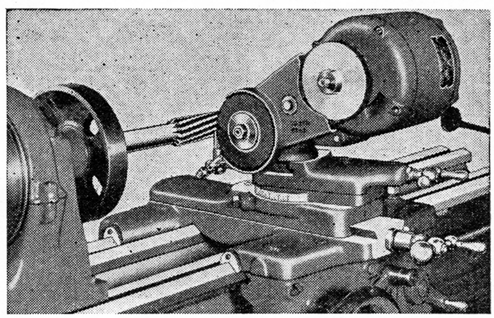


Fig. 230. Grinding a straight reamer in a lathe.

GRINDING WHEELS FOR VARIOUS KINDS OF WORK

Tabulation shows grade of Norton Grinding wheels.

Kind of Work	Rough Grind	Finish Grind
Cast Iron	3736-K Crystolon	3760-J Crystolon
Soft Steel	46-M5BE	60-M5BE
Hardened Steel	3846-L5BE	3860-L5BE
High Speed Steel	3846-K5BE	3860-K5BE
Brass or Bronze	3736-K Crystolon	3760-J Crystolon
General Work	46-N5BE	46-N5BE
Aluminum	30-M3L Shellac	36-M3L Shellac
Bakelite	3736-K Crystolon	3746-K Crystolon
Soft Rubber	3720-K5T-2 Crystolon Bakelite	3746-K5T-2 Crystolon Bakelite
Hard Rubber	3730-K5T-2 Crystolon Bakelite	3760-K5T-2 Crystolon Bakelite
Automobile Valves	1960-M	80-L6BE
Tungsten Carbide	3760/1-17 Crystolon	37100/2-H7 Crystolon

GRINDING WHEEL SPEEDS

The tabulation below shows grinding wheel speeds in revolutions per minute for surface speeds of 4000 and 5000 feet per minute.

Diam. Wheel	1 in.	2 in.	3 in.	4 in.	5 in.	6 in.	7 in.	8 in.	10 in.	12 in.
R.P.M. for surface Speed of 4,000 ft.	15,279	7,639	5,093	3,820	3,056	2,546	2,183	1,910	1,529	1,273
R.P.M. for surface Speed of 5,000 ft.	19,099	9,549	6,366	4,775	3,820	3,183	2,728	2,387	1,910	1,592

Fig. 231. Grinding wheels and speeds for various kind of work.

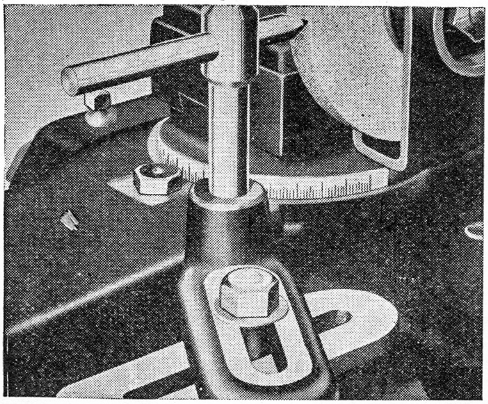


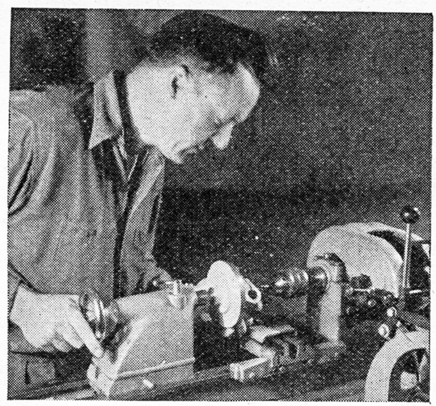
Fig. 232. Truing a grinding wheel with a diamond dresser.

removed with great accuracy and it is the quickest and best way of finishing hardened parts. Care should be exercised not to remove the material too rapidly as it will result in the part becoming overheated thus causing hardened parts to lose their temper or warp them.

The student can readily see from the foregoing lessons that the lathe is rightly termed the universal machine. The operations herein discussed in no way exhaust the possibilities of the lathe. No doubt the operator can devise many set ups not mentioned in these lessons; however, they cannot be considered standard and therefore have not been given treatment here. Likewise, much more could be said about many of the operations but the fundamentals have been covered and additional study and practice on the part of the operator is necessary before he can call himself a finished lathe operator.

In the following exercises the author presents an instructive project utilizing the foregoing operations. In making this project, the operator will have opportunity to practice the operation and care of an engine lathe. But meanwhile he should increase his skill as a lathe operator by practicing again and again the operations given here.

First Exercises for the New Operator



Before You Start Practice Work

IN THE first 43 lessons of this training course, the beginner has completed a study of the engine lathe and the operations that can be performed on it. He is now ready for practice work based on the preceding lessons.

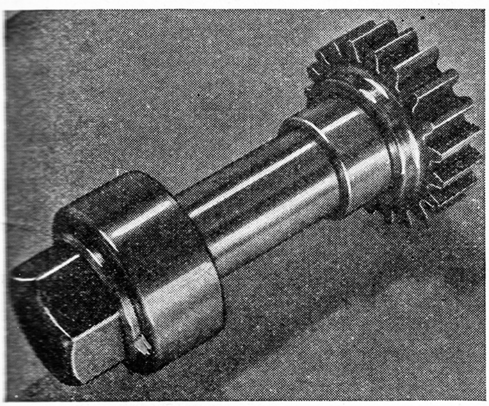
The following exercises and projects when completed within the allowable tolerances should produce a qualified engine lathe operator. These exercises have been used in the author's classes and have proved extremely valuable in training procedure. The arrangement of these exercises may be varied by the operator if he should feel that any of them are too easy or difficult for his experience.

The problems are broken down into a detailed analysis of operation sequence and procedure. The student should follow these steps closely and not vary much from the order given, but as he progresses in skill and speed he may foresee quicker, and better methods. By applying this ingenuity he can accomplish the same results with ease and precision. Whenever he feels capable of trying his hand at procedure analysis, he may try some of the problems for which only drawings are given. In this way he will have a test of his planning ability.

Before starting to work on these actual operations he should first prepare himself to operate the machine safely. These things should be part of his every-day preparation: The operator's sleeves should be rolled up above his elbows, and his tie tucked in if one is worn. Better still wear a bow tie. A knee-length blue denim apron will provide adequate protection if the operator is wearing good clothes. No jewelry, such as rings and wrist watch, should be worn while operating any machinery. All tools should be readily accessible. One of the first things to buy

is a good tool chest and then the necessary tools of good quality. The floor around the machine must be kept clear of obstacles that may cause one to trip and fall. Oil and grease spots must be cleaned up or sawdust put on them immediately so as to prevent accidents. The operator should avoid lifting heavy objects and should ask for help or use such lifting aids as are available, being sure that slings are adequate and secure. All guards must be in place before operating the machine. Such things as chuck and tool post wrenches must never be left in the socket for even a short period of time after they have performed their service.

After the operator has complied with the above regulations for his own safety, he should prepare the machine to perform its operation with safety and accuracy. Of course the machine should be lubricated as recommended by the manufacturer or on recommendations of a lubrication engineer. This is sometimes taken care of by the factory maintenance crew but the operator should know when, where and why the lathe should be lubricated. Next the lathe should be set up to perform the operation or operations called for and the first thing every good operator does is to check the live center for running axially true, then a check of both headstock and tailstock centers is made to see that they are in proper alignment, as described in Lesson 21.



Making of this project involves many of the lathe operations described in Parts 1 and 2 of this course. It is a fair test of the new lathe operator's skill.

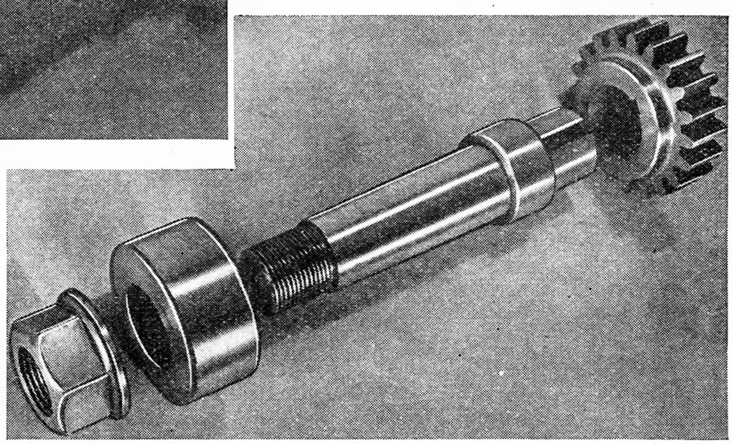
Only a small amount of ability to read blueprints is necessary for the satisfactory understanding of what is expected in producing the exercises as given herein and this can be acquired as the operator needs this information. It is best that every machinist make a study of blueprint reading as a separate course of instruction so that he will be ready for more advance work and promotions in the shop. Assuming that the student knows the bare fundamentals of reading a blueprint, he should always first study it to find out what he is expected to produce, the accuracy required, material used, whether there are any defects in the raw material, and whether there is sufficient stock for machining to specifications. If the piece part has been partly machined prior to performing his operation he should check it to see if it is correct up to that point or he may be wasting time by further machining or even be charged with spoiling the piece.

From hereon the operator applies the operation techniques as called for in the making of the following exercises and projects.

The project to be made is shown in an assembly drawing in Fig. 1. As we take up the individual operations it will be detailed.

At a glance, the student can see that the making of this project involves many of the lathe operations described in the preceding lessons.

The material used for the various exercises is either mild steel or grey iron. This may be changed by the student according to the material that he may have readily accessible but care



And here are the parts of the project, in their respective positions.

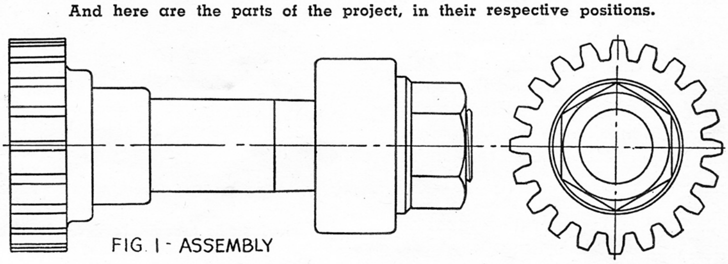


FIG. 1 - ASSEMBLY

must be exercised to use the proper speeds, feeds, tool shapes and clearance and rake angles for the material being machined.

The procedure analysis will be in outline form with reference being made as necessary to the various lessons previously given.

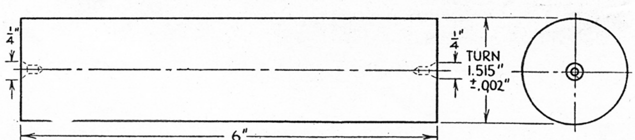
If at any time the student finds that he has gone under the allowable tolerance he should start over on a new piece. Only in this way will he learn that dimensions are important. Sometimes the piece can be salvaged, but we are not interested in this type of work.

Exercise 1—Cylinder

LESSON 44

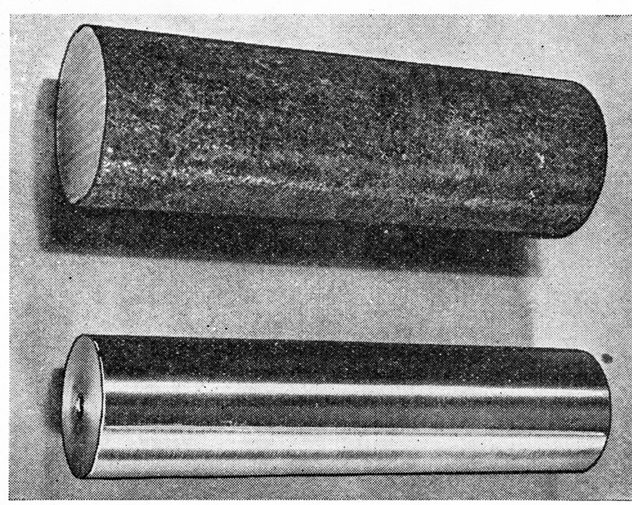
The first exercise to be attempted will be the turning of a cylinder, Fig. 2.

1. Study the drawing.
2. Secure a piece of mild steel (hot rolled preferred) approximately 2″ in diameter and 6⅛″ long.
3. Check and align lathe centers. Refer to Lesson 21.
4. Layout ends of stock, center punch and drill center holes as described in Lesson 22.
5. Mount proper size lathe dog on one end of stock and set up work between centers on the lathe. Refer to Lesson 23.
6. Select and set up correctly ground and honed tool bit in tool holder for facing. See Lessons 6, 24, and 25.
7. Calculate the correct speed to turn mild steel of 2″ diameter (r.p.m. equals C.S. × 4 divided D″). Cutting speed for mild steel may be considered as 100 feet per minute. D″ equals the diameter of the stock in inches. Solution 100 × 4 ÷ 2″ = 400 ÷ 2 = 200 r.p.m. See Lesson 5.
8. Set the feed for .012″ longitudinal travel of the table per revolution of the work. Crossfeed will be less due to gearing and is about .004″ which should give a good finish. See Lesson 5.
9. With the work mounted, the facing tool set up and the speed and feed set, one end is faced by following Lesson 26 on how to face a job between centers. Remove only about half of the excess stock off the one end, using the hand feed step method and power feed outward for the final finish cut. Lock the carriage when taking this final cut as this assures a square end on the work. Face the other end to length using caution so as to keep it within the proper tolerance on the length dimension.
10. Redrill center holes to size specified. This operation must be performed at this time.
11. Next change from a facing tool to a turning tool and make set up according to Lesson 27 for rough turning, between centers.
12. Take a light cut about 1/32 of an inch deep for about 3″ travel. "Mike" both ends of this cut to check for absolute center alignment. If there is any difference in measurements make adjustment and take additional cuts and make further adjustments until the measurements are identical.
13. Rough turn one half of cylinder allowing 1/64″ to 1/32″ for a finish cut. Note graduated dial setting. Refer to Fig. 126, Lesson 27.
14. Remove stock, place dog on other end of work and rough turn opposite end using same setting on cross feed dial. One cut if possible.
15. Increase the speed approximately 50 percent, (300 to 350 r.p.m.) reduce feed to .003″ to



FINISH ALL OVER - ENDS MUST BE FLAT
TOLERANCE 1/64" EXCEPT WHERE OTHERWISE STATED
MATERIAL - MILD STEEL

FIG. 2 - EXERCISE I
CYLINDER



Mild steel stock (above) is used for the cylinder, shown (below) after turning.

.005″ travel longitudinally per revolution. Regrind and hone tool bit if necessary or change to a finishing type of tool bit if so desired.

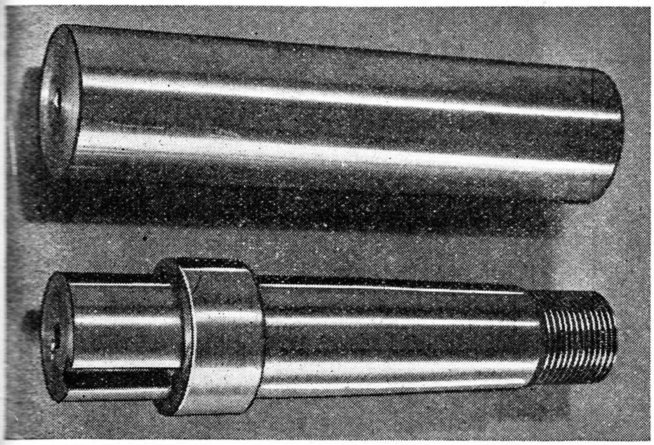
16. Take a light trial cut for about one quarter of an inch to check size and cutting action of tool according to finish and size wanted. Continue this correct cut for half the distance of the cylinder.

17. Turn piece end for end and repeat operation using same cross feed dial setting.

18. Check final piece with drawing for required accuracy. Save this piece for the next exercise.

Exercise 2—Shaft

LESSON 45



The shaft is turned from the cylinder. Compare above illustrations with plan shown below.

Use the cylinder of the previous exercise.

1. Study the drawing. Fig. 3.

2. Calculate the r.p.m.'s required for turning the various diameters. Refer to Lesson 5.

3. With the "Morphy" calipers on a sharp pointed tool and rule lay off the distances 1$\frac{1}{16}$″ and 1$\frac{13}{16}$″ (1$\frac{1}{16}$″ + $\frac{3}{4}$″) from one end.

4. Rough and finish turn the 1$\frac{1}{16}$″ distance to 1.1875″ diameter. Then shoulder the sharp corner using especially ground shouldering tool. Be sure corner is 90 degrees. See tool shapes Fig. 35, Lesson 6. See Lesson 28 also.

5. Change piece end for end and turn to 1.2105″ diameter up to the 1$\frac{13}{16}$″ layout line make in No. 3 above. Protect finish surface with a piece of copper on dog end.

6. Compute and set taper attachment or offset tailstock for correct amount of taper. Refer to Lesson 14.

7. Start turning taper, taking only light cuts until the exact taper is obtained. Refer to Lesson 37. Then proceed to turn down taper until ring gage slips on tightly 1″ from the end.

Note: A mill file may be used finally to fit the taper if only an extremely small amount of material needs to be removed. See Lesson 29.

8. Lay off on this taper a distance of 1$\frac{1}{16}$″ from the small end. Use "Morphy" calipers.

9. Turn this 1$\frac{1}{16}$″ distance down to 1.000″ diameter.

10. Neck in at shoulder $\frac{1}{8}$″ wide down to $\frac{7}{8}$″ diameter. The graduations on the crossfeed dial may be used to an advantage in obtaining the $\frac{7}{8}$″ dimension and checked after with an outside spring caliper. Refer to Lessons 8 and 31.

11. Grind tool for threading, Lesson 24. Tool must fit 60 degree angle on center gage and also national form thread gage for proper distance across flat on point for cutting 14 threads per inch. Refer to Fig. 195, Lesson 39.

12. Set the gear box or change gears on the lathe for cutting 14 threads per inch. See Les-

MATERIAL - MILD STEEL - KEYWAY $\frac{1}{8}$″ DEEP × $\frac{1}{4}$″ WIDE
FINISH ALL OVER - NOTE: FIT TAPER TO STANDARD
COLLAR TO OVERHANG $\frac{1}{16}$″ AT "B"

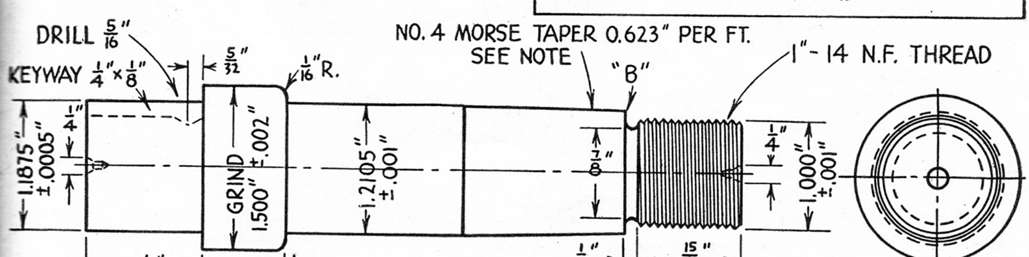


FIG. 3 - EXERCISE 2
TURNED SHAFT

son 39. Read it carefully.

13. Set up the threading tool. Refer to Lesson 24. Be sure it is set up properly.

14. Chamfer end of stock prior to cutting threads.

15. Set thread stop as explained in Lesson 39 and be sure you understand the use of the thread cutting dial, Fig. 29, Lesson 3, if one is to be used.

16. Cut threads to size following the detailed instructions given in Lesson 39. Apply thread cutting oil as needed and check the thread for size with a thread pitch micrometer as the threads approach the correct shape and size. The final test may be made with a thread ring gage or a 1"—14 N.F. standard nut.

17. Set up the tool post grinder to do external cylindrical grinding. Be sure to set work speed correctly and have the proper wheel for grinding mild steel. Refer to Lesson 43, with specific reference to Fig. 231.

18. Grind the 1.500" portion of the shaft only. Other parts of the shaft could have been ground if additional stock had been left at the time of turning and in the case of shoulder work a recess must be provided at the corner.

19. Round the external corner on the 1.500" diameter as shown. Use special form tool or fake and file to shape and dimension. Check with radius gage.

20. Layout keyway and drill 5/16" hole 5/32" deep at end near shoulder as shown in Fig. 3.

21 Cut keyway to dimensions by locking the back gears in position so the work cannot revolve and with a special tool set up in the tool holder cause the tool bit to come in contact with the work. By a hand longitudinal motion and gradual infeed the keyway can be brought to correct depth. Final fit may be made to a piece of keystock and slight filing may be necessary.

Parts made in other exercises are to fit this shaft.

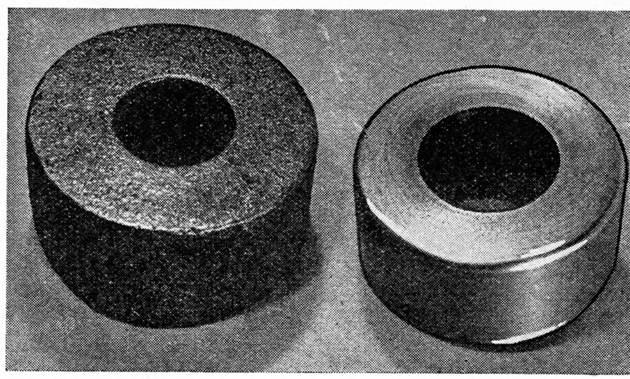
Exercise 3—Taper Collar

LESSON 46

1. Study Fig. 4 and check casting.

2. Remove fins, lumps, etc., from casting, and remove burnt core sand from walls of hole. This part may be made from a solid piece of steel if no casting is available. If steel is used calculations must be made accordingly.

3. Remove face plate and live center from lathe. Place a small ball of waste or rag in end of spindle to keep chips out. Place a four jaw independent chuck on end of spindle. Refer to Lesson 2.



If a casting is not available, the taper collar can be turned from a solid piece of steel. Study plan below.

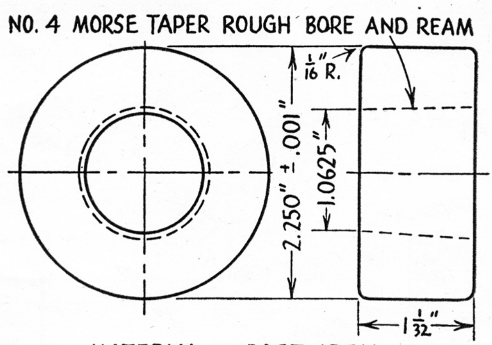
4. Place stock in chuck with small end out. Do not place piece up against face of chuck. Leave space for chips to fall out and light to get in. True piece up carefully following instructions given in Lesson 32.

5. Set up left hand round nose turning tool that has previously been ground for turning cast iron.

6. Calculate speeds for turning and boring and set machine up to operate at required speed and feed. Refer to Lesson 5.

7. Take a radial cut off exposed face. Be sure to feed from *outside* toward center. Do not remove too much, because some stock must be left for finishing other side, but be sure to get under the scale on first cut.

8. Obtain boring tool; sharpen tool bit, set up, and rough bore hole to 1 1/32" diameter. Refer to Lesson 35 on how to do boring.



MATERIAL - CAST IRON
FINISH ALL OVER

FIG. 4 - EXERCISE 3
TAPER COLLAR

9. Calculate and set compound rest to angle corresponding to taper and rough bore taper using hand feed until turned shaft, Exercise 2, will enter so that there is about $2\frac{9}{32}''$ between face of collar and the $1\frac{1}{2}''$ diameter portion of the shaft. Test taper for fit and correct if necessary before going too far. Refer to Lesson 37. Note: Taper hole may be cut by using the taper attachment if the operator so desires but for experience it is suggested that the compound rest be used. Also, if no reamer is available, the hole may be bored to exact finished size.

10. Finish ream until shaft enters, leaving $2\frac{11}{64}''$ between face of collar and $1\frac{1}{2}''$ diameter portion of shaft.

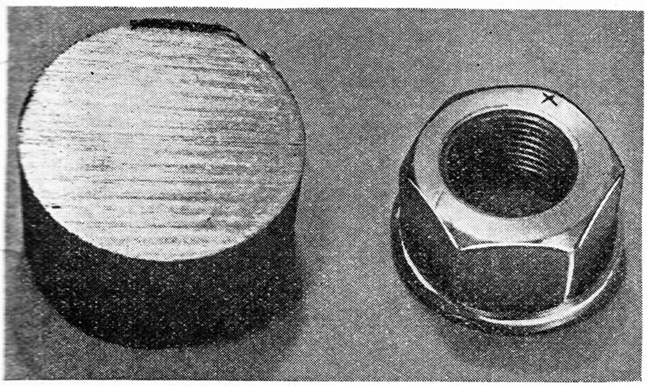
11. Remove piece from chuck, and press it tightly on the turned shaft. Use mandrel press. See Figure 174, Lesson 36. If piece does not press on so that there is $2\frac{5}{32}''$ between the face of collar and shoulder, remove shaft and hand ream hole to proper size.

12. With the taper hole properly fitted and collar pressed on shaft, proceed to turn circumference to proper diameter, using shaft for a mandrel.

13. Face other side using round nose tool until collar is proper length. Mark length with "Morphy" calipers. Collar should overhang $\frac{1}{16}''$ at shoulder "B" on shaft, Fig. 3.

14. Round corners as shown. Use special corner rounding tool or fake and file to specification and check with a radius gage.

Exercise 4—Special Hex Nut



The hex nut looks easy to make, but all nineteen operations require close attention.

LESSON 47

1. Study Fig. 5.

2. Select a piece of mild steel stock 2″ in diameter and $1\frac{1}{2}''$ in length and remove burrs from it.

3. Chuck up stock carefully in either a three or four jaw chuck. See Lesson 32.

4. Calculate the speeds for turning, drilling and boring.

5. Spot the center with a combination drill and counter sink or special center locating tool. Use cutting oil. Refer to Lesson 34.

6. Drill $\frac{1}{4}''$ hole as a pilot hole. Hold $\frac{1}{4}''$ drill in a drill chuck placed in tail stock spindle. Be sure to use cutting oil when drilling steel. Follow pilot hole with a $\frac{7}{8}''$ drill.

7. Face off not more than $\frac{1}{16}''$ off of exposed face.

8. Set up boring tool and bore hole to stated size.

9. Grind up threading tool, check it for correct shape, insert in boring bar and set up for internal threading. See Lesson 35, Fig. 165.

10. Set up lathe to cut 14 threads per inch. Refer to Lesson 39.

11. Cut threads so that threaded portion of turned shaft will just screw into special nut. This must be a good fit. Thread plug gage may be used for checking. Refer to Lesson 39.

12. Cut counterbore $\frac{3}{32}''$ deep by $1\frac{1}{32}''$ diameter as shown.

13. Remove work from chuck and place it on special nut mandrel.

14. Turn O. D. to $1\frac{3}{4}''$ diameter.

15. Face unfinished end making piece of proper length. Lay off length with "Morphy" caliper.

16. Lay off $\frac{1}{8}''$ + $\frac{3}{64}''$ from *counterbored* end.

17. Turn $\frac{45}{64}''$ distance ($\frac{7}{8}''$ + $\frac{11}{64}''$) to $1\frac{43}{64}''$ diameter.

18. Set up the milling attachment on the lathe and place a $\frac{3}{4}''$ end mill in the headstock spindle. Arrange the indexing attachment for six divisions. See Lesson 42, on how to do milling.

19. After milling, remove burrs, replace work between centers and machine the angles.

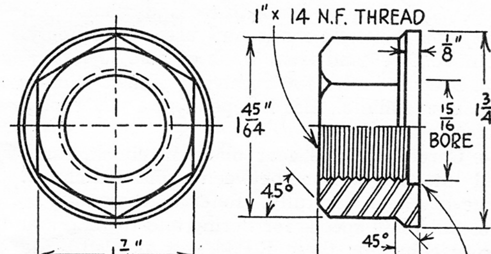
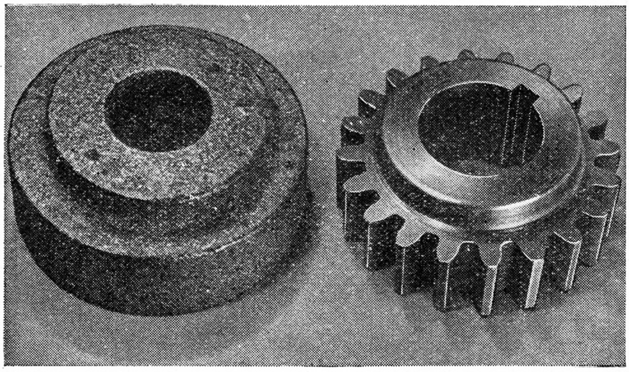


FIG. 5 EXERCISE 4
SPECIAL NUT

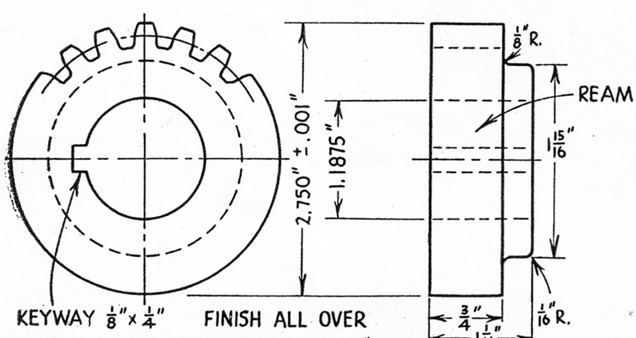
MATERIAL - MILD STEEL - FINISH ALL OVER

Exercise 5—Spur Gear

LESSON 48



If steel is substituted for a casting, in making the spur gear, calculations and tools must be made and used accordingly.



MATERIAL - CAST IRON - PITCH DIA. 2.500"
ADDENDUM - 0.125" - TOTAL DEPTH OF TOOTH 0.2696"
THICKNESS OF TOOTH 0.1963" - NUMBER OF TEETH 20
DIAMETRAL PITCH 8 - CHECK THICKNESS OF TEETH WITH GEAR VERNIER

FIG. 6 - EXERCISE 5 - SPUR GEAR

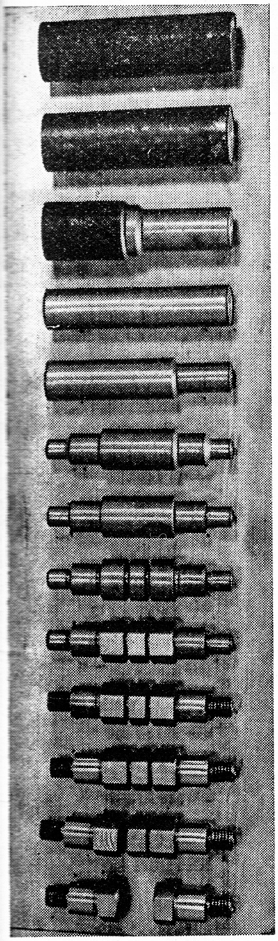
1. Study Fig. 6.
2. Remove fins, lumps, etc., from casting. A piece of steel may be substituted and calculations and tools must be made and used accordingly. Remove core sand from walls of hole.
3. Prepare lathe for receiving chuck. Screw chuck on spindle. Use independent 4 jaw chuck. Refer to Lesson 2.
4. Carefully place gear blank in chuck, small end out. Leave space between blank and chuck. True blank up carefully. Refer to Lesson 32.
5. Calculate speeds for boring and turning and set machine for these R.P.M.'s as needed. See Lesson 5.
6. Use round nose turning tool. Face off exposed face, being careful to leave enough stock for finishing opposite side. Be sure to get under the scale on the first cut.
7. Set up boring tool and holder. Bore hole to 1.1775". Refer to Lesson 35.
8. Place a $1\frac{3}{16}$" (1.1875") machine reamer in tailstock spindle or special floating reamer holder and ream hole to size. Speed should be reduced for reaming. Push reamer through rapidly. Test hole with plug gage. Refer to Lesson 34. If no reamer is available hole can be bored to size.
9. Cut internal keyway to size given in manner similar to the method used in producing the external keyway on the shaft, only use a boring bar to hold the special tool.
10. Remove blank from chuck and place on mandrel. Use drop of oil on mandrel. Use mandrel press. Refer to Lesson 36.
11. Place work between centers and finish turn blank to proper dimensions.
12. Face unfinished side of blank, making blank to proper thickness and fillet to proper size. ($\frac{1}{8}$" radius.) Also round corner to $\frac{1}{16}$" radius as shown. Slow down lathe speed for these operations. Use form tools on radii turning or fake. Check all radii with fillet or radius gages.
13. The blank is now ready to have the teeth formed on the circumference. Set up the lathe with the gear cutting milling attachment in a manner similar to that used in milling the special hex nut. Select the proper cutter and mount it on an arbor held between centers and cause it to rotate at the proper speed. (Same formula may be used in figuring required R.P.M. to rotate cutter but instead of using the diameter of work, the diameter of cutter is used. Be sure to note whether the cutter is made of carbon or high speed steel.) After two teeth are cut, check with gear tooth vernier. See Fig. 54, Lesson 11.
14. Mill the rest of teeth per specification as shown on drawing. Two cuts, a roughing and finishing cut, may be required in order to produce a good finish and maintain the required accuracy.

The finished exercises may now be assembled as shown in Fig. 1 and the project is complete.

While this project is not a useful tool, it does involve many lathe operation experiences and is therefore excellent as a training device.

Special Bolts—Lathe Center—Crankshaft

LESSONS 49, 50 and 51



For additional practice and experience, make these special bolts.

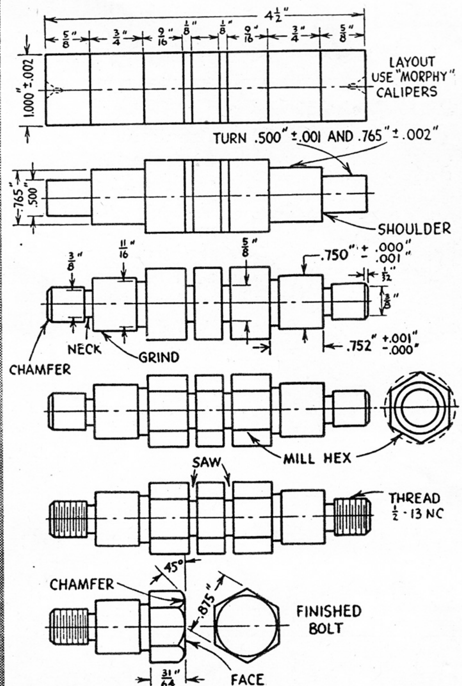
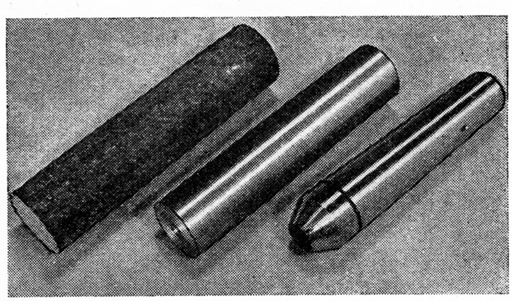
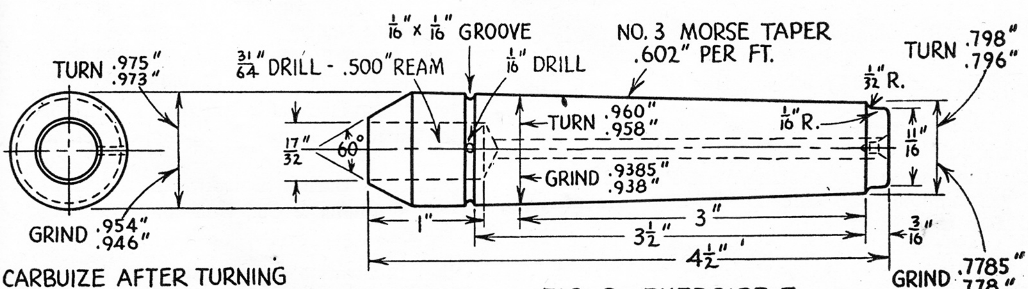


FIG. 7 - EXERCISE 6
SPECIAL BOLTS

For additional practice and experience, the student may attempt the making of the special bolt exercise shown in Fig. 7. First study the drawings and jot down the procedure to follow. The making of these special bolts should be relatively easy as the pictures indicate the step by step procedure. In this exercise, the student will learn how duplicate parts may be turned out and time saved by performing the various operations first on one end and then by reversing the work on the centers and using the same tool and tool setting the other end is machined to like size and shape. The average time for the beginner to complete this exercise should not be in excess of nine clock hours. Of course if many such special bolts had to be produced, the job would probably be made on a turret lathe or automatic screw machine using hexagonal shaped bar stock. In the latter case a finished bolt would be made in approximately five minutes.

The lathe center (below) will offer additional taper turning experience.





CARBUIZE AFTER TURNING
HARDEN - ROCKWELL C60-2 GRIND TO SIZE
INSERT - HIGH SPEED OR CARBIDE

FIG. 8 - EXERCISE 7
LATHE CENTER

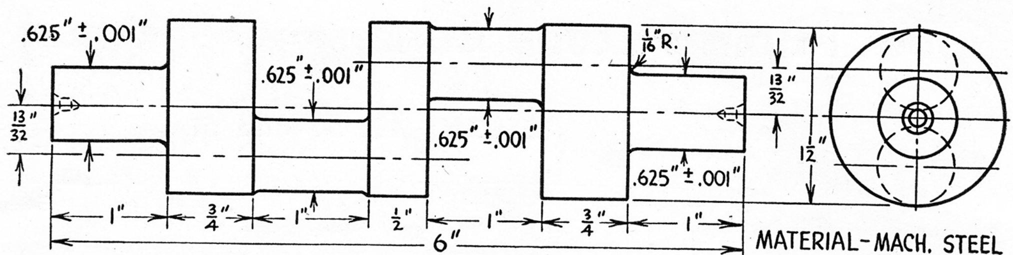
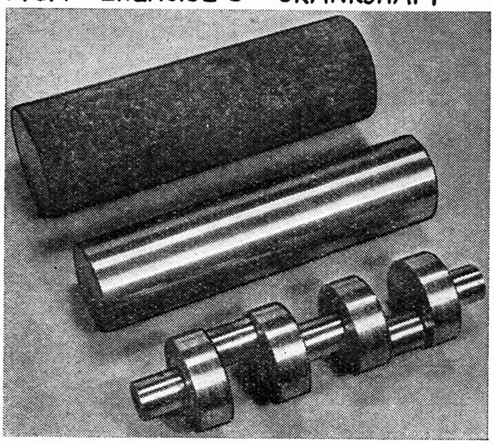


FIG. 9 - EXERCISE 8 - CRANKSHAFT



And for an extra test of one's lathe skill, try making this crankshaft.

The lathe center, Fig. 8, will offer additional taper turning experience and will also give the student an opportunity to do some heat-treating. The shape and size might even be changed to meet the need and material at hand. That is, a soft live center may be made for the student's own lathe or tool steel may be used for a dead center. In either of these cases the stock would necessarily be longer as the point would be included as part of the finished project. Also as an added feature the student may try his hand at eccentric turning by making the small crankshaft shown in Fig. 9.

After completion of the above course of instruction the student lathe operator should possess enough experience and confidence to tackle almost any small job. Continued study and practice will soon make a qualified lathe operator capable of commanding a good position with the many advantages that are possible in this field of work.

Thank You, Mr. Manufacturer

SCIENCE AND MECHANICS wishes to thank the manufacturers listed below for their helpful advice and assistance in the preparation of this course of lessons on "Operating an Engine Lathe." They have generously supplied information, technical data, drawings, and photographs. These manufacturers are as follows:

Armstrong Bros. Tool Co., Chicago, Ill., Figs. 12, 13, 14, 15, 16, 17, 18, 140, 142.

Atlas Press Company, Kalamazoo, Mich., Figs. 3, 31, 46, 104, 159-B, 160-B, 163, 165, 174, 179, 193.

Brown & Sharpe Mfg. Co., Providence, R. I., Figs. 48, 49, 51, 52, 53, 54, 56, 57, 82, 202.

The Carborundum Company, Niagara Falls, N. Y., Fig. 137.

The Cleveland Twist Drill Company, Cleveland, O., Figs. 58, 61, 62, 78.

The Cushman Chuck Co., Hartford, Conn., Fig. 19.

Henry Disston & Sons, Inc., Philadelphia, Pa., Fig. 74.

The Dumore Co., Racine, Wis., Figs. 227, 228.

Greenfield Tap and Die Corp., Greenfield, Mass., Fig. 76.

Heller Brothers Co., Newark, N. J., Figs. 75, 136.

The R. K. Le Blond Machine Tool Co., Cincinnati, O., Figs. 6, 23, 24.

Logan Engineering Co., Chicago, Ill., Fig. 35.

The Lufkin Rule Co., Saginaw, Mich., Figs. 36, 38, 39.

The Monarch Machine Tool Co., Sydney, O., Fig. 7.

Morse Twist Drill & Machine Co., New Bedford, Mass., Figs. 59, 60, 63, 64, 65, 66.

Norton Company, Worcester, Mass., Fig. 231.

Sheldon Machine Co., Inc., Chicago, Ill., Figs. 5, 22.

The Skinner Chuck Company, New Britain, Conn., Fig. 20.

South Bend Lathe Works, South Bend, Ind., Figs. 4, 25, 26, 27, 28, 29, 30, 32, 33, 148, 158, 159, 160, 164, 167, 180, 184, 187, 188, 189, 190, 194, 195, 196, 197, 216, 217, 218, 223, 224, 225, 226, 230, 232.

The Standard Tool Co., Cleveland, O., Fig. 21.

The Walton Company, Hartford, Conn., Fig. 77.

The Warner & Swasey Company, Cleveland, O., Fig. 8.

Whitman & Barnes, Detroit, Mich., Figs. 97, 98.

J. H. Williams & Co., Buffalo, N. Y., Figs. 9, 10, 11, 204.

Other illustrations not listed above were made with the co-operation of the Northwestern Technological Institute, Evanston, Ill.

TECHNICAL BOOKS
TO HELP YOU EARN MORE

Brush up on the math you've forgotten—learn new trades—improve your skill on special operations—put your latent abilities to work—for your own good and for your country. These books have been selected as the best books in their particular fields. Check the list now and order the books you need. Let these books help you get an increase in salary. Clip and mail the coupon—NOW!

SIMPLIFIED RADIO SERVICING. New comparison method enables you to locate source of trouble in any radio without equipment. Diagrams, charts, tables and graphic illustrations explain exactly how to proceed, 112 pages, 16 large trouble-shooting blueprints, over 1,000 repair hints. Covers every radio set—new and old. Learn to repair radios in minutes instead of hours. $1.50.

AMERICAN MACHINISTS' HANDBOOK AND DICTIONARY OF SHOP TERMS. A standard manual of data, methods, and definitions for apprentices, machinists, toolmakers, draftsmen, designers, foremen and all others connected with shop and drafting room practice in the metal-working and allied industries. 1,387 pages. $4.00.

MACHINERY'S HANDBOOK. World-renowned book for mechanical engineers, machine designers, toolmakers and machinists. Contains engineering and manufacturing standards, machine shop data, rules, formulas, and general information for designers and builders of everything mechanical. 1,815 pages. Illustrated. $6.00.

EXPERIMENTAL ELECTRONICS. This is the first book dealing with the noncommunication uses of electron tubes. It contains previously unpublished methods and techniques. 70 experiments illustrate principles and applications. 177 diagrams. $4.65.

STANDARD HANDBOOK FOR ELECTRICAL ENGINEERS. The great standard reference work in the field of electrical engineering practice, now in its 7th edition. 2,303 pages of the latest information on electricity and its applications. $8.00.

MECHANICAL ENGINEERS' HANDBOOK. Fourth edition of this well known handbook. Much new material has been added covering the theory of models, wind pressure on structures, powder metallurgy, etc. Gives you the facts quickly. 2,250 pages of data. $5.00.

AIRCRAFT POWERPLANT MANUAL. A practical instruction and reference book on all phases of aircraft powerplant construction, operation, maintenance, servicing and repair. Includes mathematical tables. $4.00.

GAGES AND THEIR USE IN INSPECTION. Complete study of the use of gages, pointing out why gages are made, and describes how they are used in a variety of work. Promotes more efficient use of gages in the shop. $1.50.

MODERN RADIO SERVICING. Complete, 1300-page, 1-volume course in radio repair work. Explains every phase of the entire art of servicing and repairing radio equipment by the most advanced methods. All test instruments, troubleshooting and repair procedures explained in detail. $5.00.

BLUEPRINT READING FOR THE METAL TRADES. Enables mechanics to read prints and free hand sketches with ease. A very clear handbook showing blueprints as they actually appear. $2.75.

A PRIMER OF BLUEPRINT READING. A simple and elementary text enabling the beginner to read and thoroughly understand blueprints. Paper bound. 60c.

INDUSTRIAL PLASTICS. Wartime developments in plastics, including their use for planes and their parts, bomb fins, bullet-proof composite plates, submarine battery holders, machine gun carriers, cartridge shells, infantry gun stocks, etc., are the features of this standard text on plastics. Many suggestions for new applications and markets are helpful to those associated with the production and fashioning of plastics as well as to manufacturers. 385 pages, heavily illustrated with halftones, charts, engineering data, and plates in full color. $4.50.

ENGINEERING DRAWING. A new, distinctive text which teaches effectively fundamentals, accurate reasoning, techniques, and applying theory to practice. Qualifies the user for advanced design. 400 pages, illustrated. $2.75.

HANDBOOK OF CHEMISTRY AND PHYSICS. Over 2,500 pages of data on all phases of chemistry and physics, including latest research data. Included also are complete mathematical tables. $4.00.

HANDBOOK OF MECHANICAL DESIGN. Contains a vast fund of engineering data and many practical methods of design procedure not to be found in ordinary handbooks. $4.00.

SHEET METAL WORK. Practical manual of self-instruction in pattern drafting and construction work in light and heavy gauge metal, including skylights, roofing, cornice work, patterns for forced air fittings. 360 pages. $2.50.

BLUEPRINT READING FOR THE MACHINE TRADES. Takes up principles of working drawings, relation between objects and mechanical drawings, definitions of machine shop terms, reading working drawings, assembly, scale and working drawings, etc. 154 pages, 92 illus. and tables. $1.50.

ESSENTIAL MATHEMATICS FOR SKILLED WORKERS. Simplified presentation of arithmetic, algebra, geometry, trigonometry, and slide rule as applied to problems of the machine shop, sheet metal shop, radio and airplane industries. $2.00.

MECHANICAL MOVEMENTS. Valuable to inventors, mechanics, and artisans. 256 pages of fundamentals on which all machines are built; methods of changing rotary into rectilinear motion, rectilinear into oscillating motion, slow into fast motion; hundreds of other principles of modern machinery. Contains photographs of working models demonstrating these principles. $1.98.

RUNNING A MACHINE SHOP. Valuable pointers for more efficient and profitable operation. Covers everything from shop layout to inspection methods; how to equip the shop, route and handle work, train workers, etc. 412 pages. $3.50.

ENGINE LATHE OPERATIONS. 104 lessons. 10 jobs in blueprints bound in this book. 310 illustrations show details of lathe operations, attachments and tools used in lathe work. 150 pages. $1.98.

FIRST PRINCIPLES OF RADIO COMMUNICATIONS. Concise, readable summary for beginners gives basic understanding of principles that make each part of a radio transmitter and receiver work—and gradually develops the subject of radio practice. 366 pages. 183 diagrams. $3.00.

MACHINE TOOLS AT WORK. 430 illustrated descriptions of operations on many different types of modern machine tools. Operations show practical applications of successful advanced methods and principles. 522 pages. $4.00.

PLASTICS FROM FARM AND FOREST. 157 pages about plastics today; their origin in cotton, wood, lignin, soybeans, farm wastes, cloth, skim milk, water, oil, coal, air, and rubber. Tells what plastics are; where they come from; how they are used in peace and war. $2.00.

RADIO PHYSICS COURSE. Adapted for home study or for beginner. Gives information on the working of radio receivers, photoelectric cells and devices, sound amplifier equipment, television equipment, and sound motion pictures. 972 pages, 508 illus. $5.00.

PRACTICAL MATHEMATICS FOR HOME STUDY. Step-by-step treatment with answers of the essential mathematics for the trades, shop, drafting and other jobs. Up-to-date exercises and problems. Arithmetic, geometry, algebra, and trigonometry are thoroughly covered. 697 pages. $4.00.

Send Your Order On This Coupon

SCIENCE AND MECHANICS MAGAZINE,
154 East Erie Street, Dept. L-244, Chicago, (11).

Enclosed find $.............. for which send me the following books, all charges prepaid.

....

NAME ....

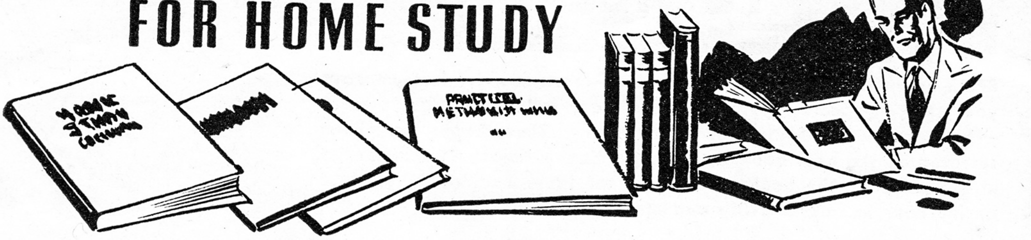
STREET & NO. ....

CITY & STATE..

☛ If you prefer, books sent C.O.D., plus postage in U. S. only.

PRACTICAL BOOKS

FOR HOME STUDY



ESSENTIALS OF UPHOLSTERY. Gives complete description of all the tools, processes, and materials necessary for the upholstering of everything from the padded seat of a stool, to an overstuffed davenport. Teaches tacking and stretching webbing, tying springs, cutting covers, etc. $1.75.

INTERIOR ELECTRIC WIRING AND ESTIMATING. Covers all phases of interior wiring including estimating of labor and materials. 354 pages, 332 illustrations, 41 tables, 8 full-sized blueprints of typical houses for estimating. $2.50.

MODERN ELECTRIC AND GAS REFRIGERATION. Discusses fundamentals of refrigeration; how to install, service, and repair refrigerators of all types. 928 pages, 408 illustrations and charts, many in color. Outstanding work in this field. $5.00.

ARMATURE AND MAGNET WINDING. Explains proper procedure in winding all types of armatures and magnets. Illustrations, charts, and tables show exactly what to do and how to do it. 260 pages. $1.50.

PRACTICAL ELECTRICITY AND HOUSE WIRING. This new and up-to-date book discusses these very important subjects: Dictionary of common electrical devices, basic principles of circuits, wires and insulation, National Electrical Code, three-wire system, grounding, polarizing, fuses and circuits, wiring methods, service entrance, planning your installations, wiring new houses, installing switches, fixtures, etc., wiring houses after they are built, alterations and additions, wiring farm buildings, wiring for motors, AC and DC, single-phase and three-phase, home repairs, etc. 190 pages, 208 illus. $1.50.

ODD PIECES IN WOOD. Useful articles you can make from scraps. Among the projects are a collapsible table, auto creeper, typewriter table, shoe cabinet, tray table, cutouts, etc. 28 working drawings. 65c.

YOU CAN FIX IT. Up-to-date encyclopedia of home repairs and household hints. Covers everything from cellar to roof and all the appliances found in the home. Tells you what to do if your cellar is damp; if your furniture needs re-upholstering; if your heating system breaks down; if your iron won't work; if your radio or plumbing "acts up," and hundreds of other useful facts that will save you money. 480 pages. 75c.

SLIDE RULE AND LOGARITHMIC TABLES. Explains slide rule principles; use of the slide rule, polyphase slide rules; circular slide rules; calculations involving angles; using tables of logarithms, etc. 200 pages. $1.50.

HOME OWNER'S HANDBOOK. Authentic information on selection of building site; financing; landscaping; water supply; plumbing and heating; lighting appliances; insulation; modern kitchens and bathrooms; playroom; use of tools; and hundreds of hints for improving your home. 197 pages, 173 illustrations. 75c.

24 ACTION TOYS. Complete handbook of toy-making. Covers tools and supplies; transferring designs; sawing; fastening parts together; coloring and finishing; and design for making standing, rocking, and wheeling animals including kangaroo, bear, camel, tiger, buffalo. Also designs for jumping jacks, acrobats, pecking hens, shimmy walker, bronco buster, etc. 60c.

BLUEPRINT READING FOR THE BUILDING TRADES. An up-to-date book for beginners and experienced building men. Shows how to read and understand construction blueprints as you would a newspaper. Every man in contracting for the building trades should read and study this fine book. 111 pages, 69 illus. $1.25.

ROPPS COMMERCIAL CALCULATOR. Short cut arithmetic for addition, subtraction, multiplication, and division. Quick methods for figuring interest, wages, measurements, costs, discount, taxes, etc. Scores of tables for instant figuring on hundreds of subjects. Cloth, 50c; Kraft Leather, $1.00.

STEEL SQUARE. Complete explanation for using this important tool. Covers various scales, using squares, calculating lengths, laying out rafters, figuring roof spans, runs, rises, pitches, etc. A real "how-to-do-it" book enabling you to learn all standard uses of the steel square. 101 illustrations. $1.25.

CARPENTRY. This practical 436-page book explains principles of building construction, including rafters, roofs, floors, walls, studding, girders, partitions, bridging, trusses, etc. How to make all types of joints, how to build concrete forms, how to finish both inside and outside, and hundreds of other important subjects. 541 illustrations. Exceptional value. $2.00.

FOUNDRY WORK. Discusses the fundamentals of molding practice, processes, methods, materials, fuels, sand handling, pouring, metallurgy, heat treating, etc. 200 pages, 140 illustrations. $2.00.

ART OF WHITTLING. Whittling toys and queer things with a good pocket knife offers hours of entertainment and pleasure. This book gives detailed instructions and diagrams for whittling puzzles, chains, pliers, scissors, fans, toys, cage and ball, building things in bottles, etc. $1.50.

ARMATURE WINDING. Explains with 246 illustrations, connecting diagrams and detailed instructions how to wind, test and repair A.C. and D.C. motors and generators. Written especially for home study. $2.00.

MIXING OF COLORS AND PAINTS. Practical information on art of mixing colors and paint for interior decorating, house painting, arts and crafts, color formulas, color cards, furniture, polychrome finish, selecting color schemes, thinners, dryers, etc. 296 pages. $2.00.

ELECTRICAL TROUBLE SHOOTING ON THE MOTOR CAR. Explains fundamentals of wiring, generators, starters, battery ignition, magnetos, storage batteries, coil testing, voltage and current regulators, armature and field testing, rewinding armature and field coils, tuning up the motor, etc. In all, 567 pages, 58 chapters, over 350 illustrations. $4.00.

ELECTRIC MOTOR REBUILDER'S GUIDE. Tells how to recondition all types of popular AC electric motors and controllers, explains automatic winding of armature coils, includes many helpful rewinding charts. $1.50.

HANDBOOK FOR PROSPECTORS. Tells what to do to locate valuable minerals; how to identify, sample, test, appraise, mine and market minerals of all kinds. $4.00.

ELECTRIC MOTOR TROUBLE SHOOTER'S GUIDE. Tells how to locate and correct all ailments of DC electric motors and controlling devices. Richly illustrated with explanatory charts and diagrams. $1.50.

HOW TO RAISE RABBITS FOR FOOD AND FUR. Complete instructions and details of rabbit culture, the selection and purchase of stock, exhibiting, selling, etc. Raise rabbits to replace rationed meats. 256 pages. $2.00.

2100 NEEDED INVENTIONS. Source book of practical money-making ideas that are crying to be invented. Special chapters for wartime inventors, on the art of inventing, and how to develop and protect ideas. 252 pages. $2.00.

PLUMBING INSTALLATION AND REPAIR. 236-page book explaining how to do repair jobs such as leaky faucets, clogged toilets, drains, traps, vents, sewers, etc. Tells how to cut pipe, make pipe joints, and install fixtures. 153 illus. $1.50.

THINGS TO MAKE IN YOUR HOME WORKSHOP. Plans for making toys, sailboats, lamps, pirate's chest, mirrors, cabinets, clocks, bookshelves, chairs, etc. Explains how to make joints, do inlay work, wood turning, veneering, etc. 256 pages. $1.00.

PLASTICS—PROBLEMS AND DESIGNS. Shows how to design and make scores of practical articles such as lamps, clocks, jewelry, ash trays, etc., from cast resin plastics with ordinary tools. 233 pages. $2.50.

USE THIS HANDY COUPON NOW

SCIENCE AND MECHANICS,
154 East Erie Street, Dept. L-144, Chicago (11).

Enclosed is $................................for which send me the following books, all charges prepaid.

....

NAME ....

STREET AND NUMBER..

CITY AND STATE..

☛ If you prefer, books will be sent C. O. D., plus postage in U. S. only.